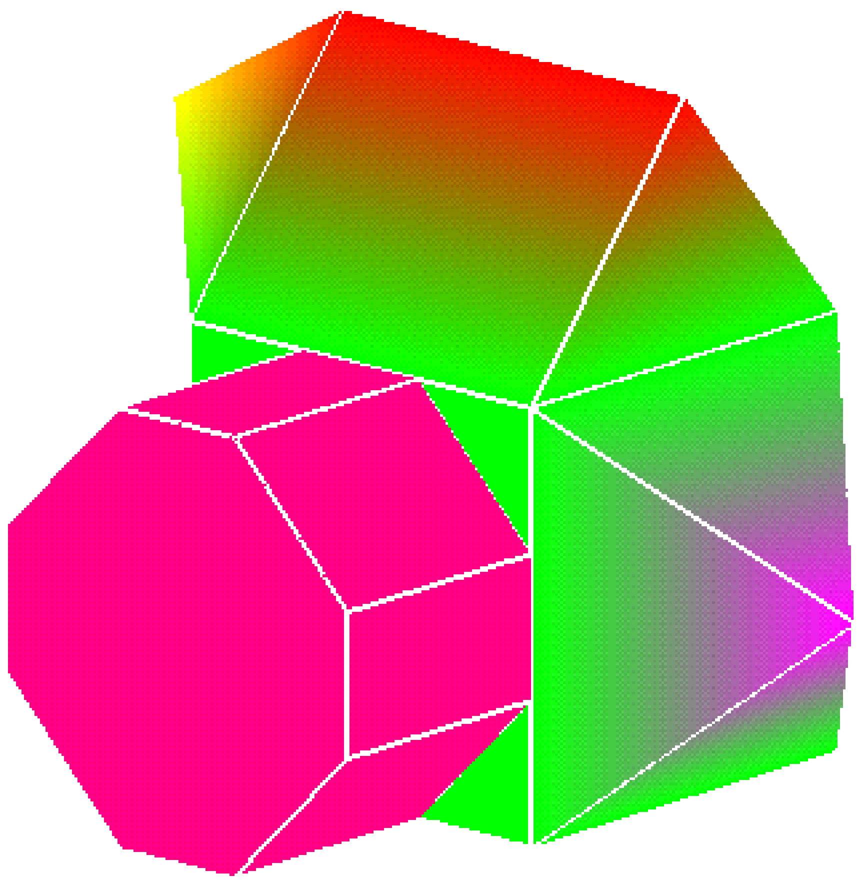
LAUR 95−2986
*Manual*

# GMV *Version 0.9.8*

## ***General Mesh Viewer***

## *User's Manual*




By Frank A. Ortega

**Los Alamos**

**NATIONAL LABORATORY**



*Los Alamos National Laboratory is operated by the University of California for the United States Department of Energy under contract W−7405−ENG−36*

*Edited by Patricia W. Mendius, Group CIC−1*

# Table of Contents

# Illustrations Listing

# Preface

## Revision Record for GMV:

| Revision/Date: | Description: |
| --- | --- |
| July 1997 | Version 0.9.8 |

## Description and Intent:

This manual describes the General Mesh Viewer (GMV).

GMV is a three−dimensional visualization tool that can process data from any 3−D mesh. Data to be visualized are taken from a properly formatted input file and displayed on the screen. With simple pull down menus, windows, and mouse controls, many special functions are available to maximize the practical value of any simulation GMV may be asked to visualize.

This manual has three purposes: to teach the beginner how to operate in the GMV environment, to act as a reference guide for the more experienced user, and to define the format for the GMV input file.

The only possible prerequisite for the use of GMV is the knowledge of a computer programming language so that you can write code to generate GMV input data. However, finished input files are not very difficult to obtain and can also be written manually using a text editor.

## Syntax Conventions:

Words enclosed in double quotes ("like this"), unless otherwise stated, are actual quoted material from the GMV environment, such as menu options or error messages. This punctuation does not apply to the description of the GMV input format. The GMV input file section has its own set of conventions.

# *Getting to Know GMV*

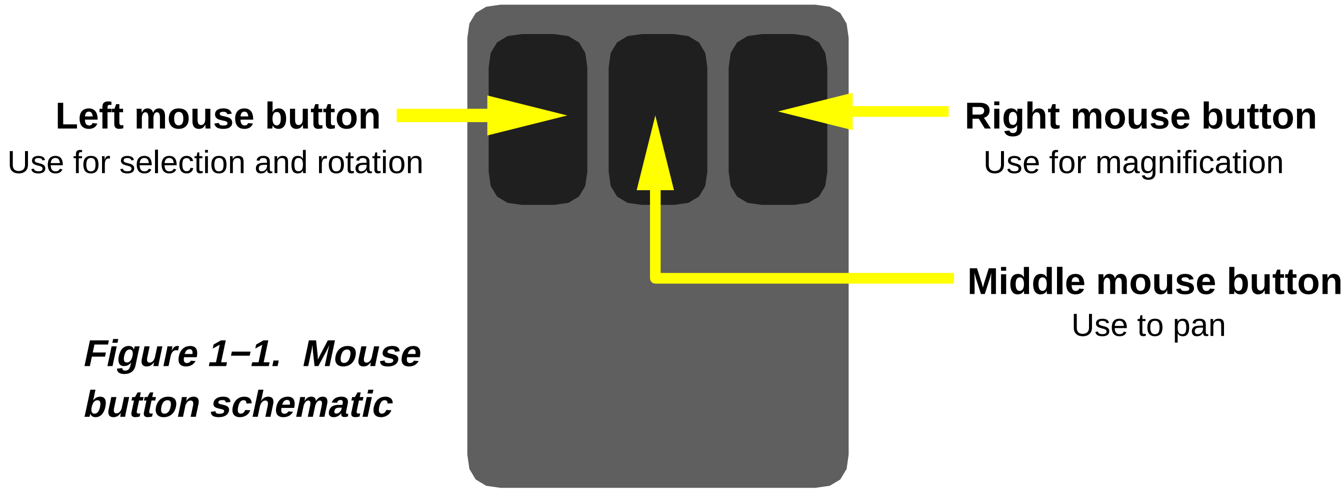



***Figure 1−1. Mouse button schematic***

## Starting GMV:

To start GMV, the location of the executable must be known. If GMV is in the current path, simply type:

**`gmv`**

on the console and press the enter key. The main GMV window will appear first, followed by a File Selection menu requesting the name of a GMV input file. Double click on the name of an input file, or select the file and click on "OK." If the requested file is not a valid GMV input file, then a box stating this fact will appear and allow you another chance to select a valid GMV input file. After the file is chosen, the mouse pointer will change into a watch, indicating that GMV is processing the input file and preparing to display the data on the screen. Be patient, this may take a few seconds or even minutes depending on the size of the input file. An object will then be displayed on the screen. The object displayed depends on the input data and the following order: polygons, cells, nodes. If polygons exist, they are drawn first. If no polygons exist, then cells are drawn. If no cells exist, then nodes are drawn, which must exist in a valid input file. In addition, the "Display" window corresponding to whatever was displayed first will pop up. For example, if the input file contains polygon data, the "Polygons" window will automatically pop up when the input file is first opened.

## The GMV temporary field files:

Upon the opening of any input file, GMV creates temporary files on the local system which hold all of the node and cell field data read in from the input file. The first three characters of these files are "GMV" which are followed by additional characters. GMV will first attempt to place this file in the directory specified in the environment variable "TMPDIR." Set this directory with the C shell command:

**setenv TMPDIR *directory_name***

where *directory_name* is the path where you want GMV to place the field data file. If "TMPDIR" is undefined, GMV will attempt to put the field data file in "/usr/tmp." These temporary files are removed upon normal completion of GMV.(Choosing "Quit" from the file menu)

## The GMV user interface:

### The mouse controls

The mouse controls for GMV have been designed for maximum ease of use with a three button mouse, (see Fig. 1−1). The left mouse button has two different functions, depending on whether the cursor is in a menu area or the viewing area. In the case when the mouse pointer is in the menu area, the left mouse button is used to pull down menus, select options, drag slider bars, etc. When the mouse pointer is in the viewing area, the left mouse button functions as a rotation device. For example, (assuming twist is set to zero) while holding down the left mouse button and dragging left or right, the object in the viewing area rotates either left or right, depending on the current orientation of the axes. Moving the mouse up and down in this manner rotates the object either up or down, again depending on the current placement of the axes. The middle mouse button provides a panning function. Holding the middle button and moving the mouse shifts the object linearly in any direction without any rotation. For example, while holding down the middle mouse button and dragging right, the object moves to the right. Finally, the right mouse button is used for controlling the magnification of the object in the main viewer. For example, while holding down the right mouse button and dragging up, the object grows larger. Dragging down causes the object to appear smaller. Motion to the left or right does nothing.

### Using menus

The top row of the GMV window is lined with various menus. To open a menu, click the left mouse button on the name of the menu desired. A small box will appear with menu options. To select a menu option, again click the mouse on the desired option. Some of the menu options will open sub menus for specific program functions which require additional information. To choose from a submenu, click on the original option and move the mouse to the right and follow the usual rules for choosing from menus.

## Buttons

GMV has three different types of buttons which are used to select various functions: regular buttons, toggle buttons, and radio buttons. Regular buttons are fairly large and have labels inside such as "CLOSE" or "CANCEL." To activate these, just click the mouse on the button desired. It will temporarily depress to indicate that it has been activated. The toggle buttons used in GMV are small and square in shape with labels next to them. These buttons have two different states, on or off. The little indented square will appear yellow when it is on, and grey when it is off. Radio buttons are a set of toggle buttons that allow only one selection of the set and are diamond shaped.

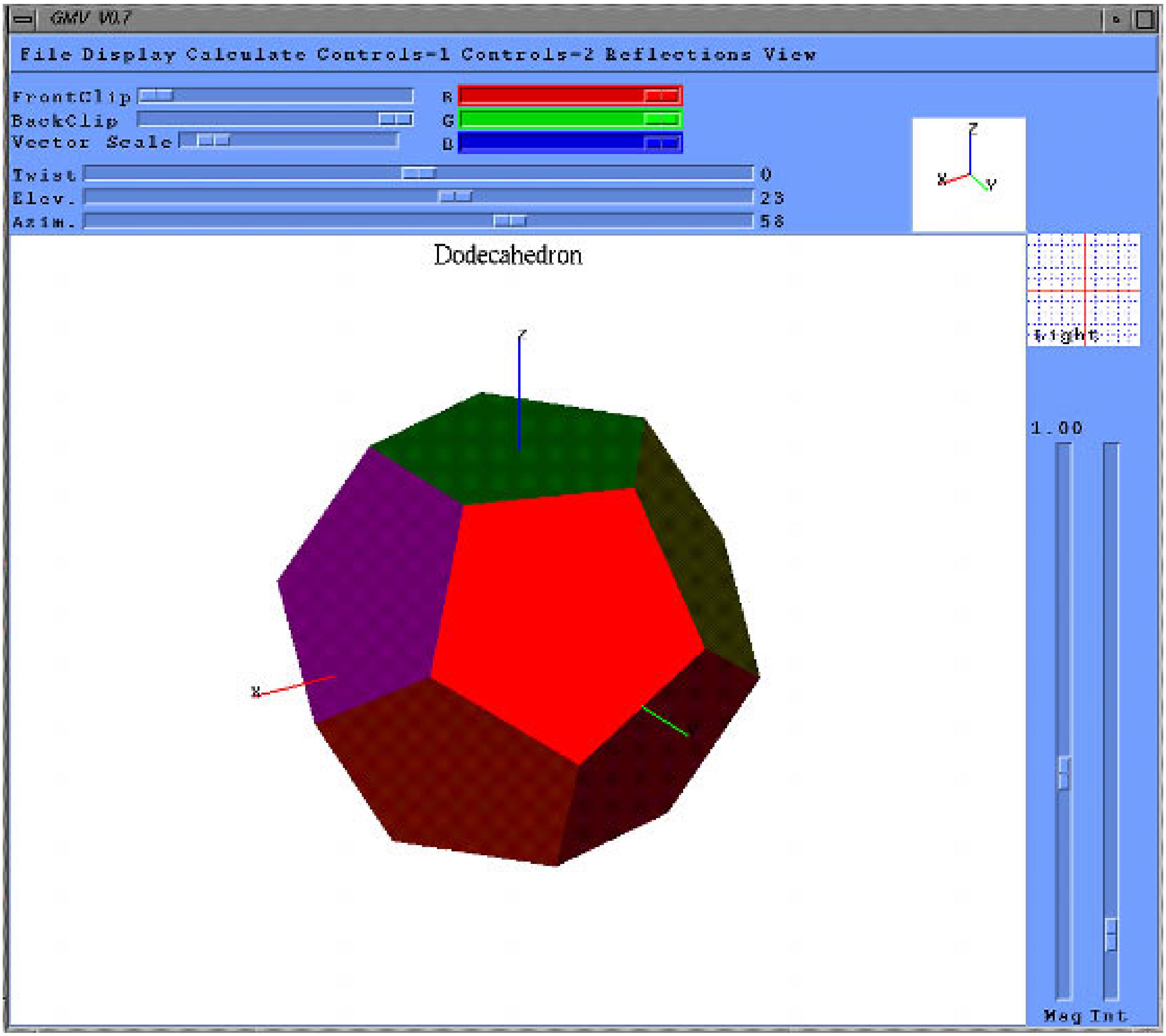



***Figure 1−2. Main GMV window***

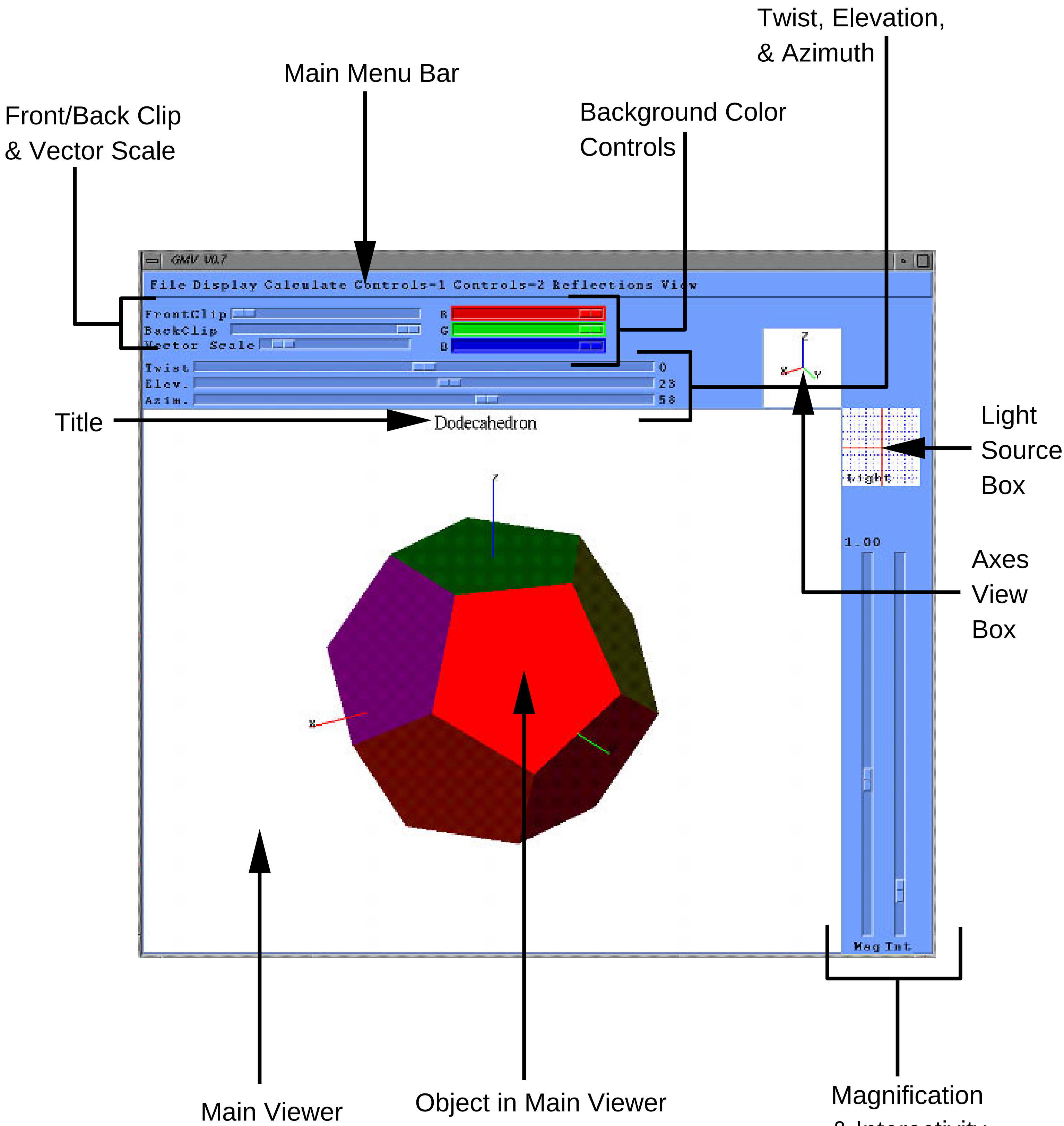



***Figure 1−3. Main GMV window cont.***

### Slider bars

Slider bars are control devices used throughout the GMV user interface. The use of slider bars is very easy. Just click and hold the left mouse button on the rectangular shaped slider control and drag it back and forth until the desired adjustment has been made. When more precise changes are desired, you can click the left mouse button on a portion of the slider bar not covered by the slider control and the slider will move at predetermined units.

### Scroll bars

Many menus within the GMV environment contain lists of things to choose from. Lists are placed into scroll boxes. On the right side of a scroll box is the scroll bar. The scroll bar is much like a slider bar. To scroll through a list, click and drag the slider back and forth in its track until the desired part of the list is in view. You may also click in the scroll bar's track on either side to move through the list more slowly.

## The main GMV window: (see Fig. 1−2 & 1−3)

### The menu bar

The menu bar is located at the very top of the main GMV window. The

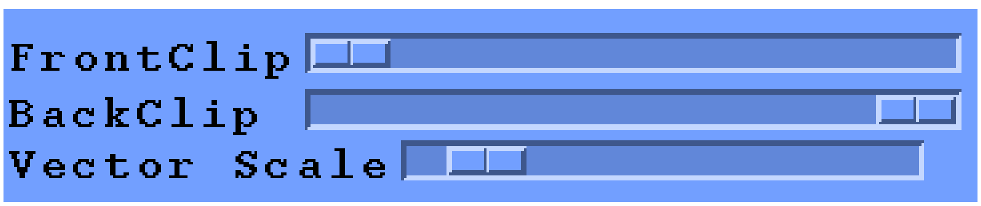



***Figure 1−4. Various slider bars***

menu names listed in order are: file, display, calculate, controls−1, controls−2, reflections, and view.

### FrontClip/BackClip

Think of frontclip and backclip as moving planes that erase everything in their paths, (see Fig. 1−4). Frontclip starts with an invisible plane parallel to the screen and in front of the object in the main viewer. Dragging the slider bar to the right moves this plane forward, clipping the object as it goes. By the time the slider bar is all the way to the right, there will be nothing left to view. Backclip is just the opposite. Backclip starts with an invisible plane parallel to the screen and behind the object. Dragging the slider bar to the left moves the plane toward you, again clipping the object as it moves. When the slider is all the way to the left, none of the original object will remain. When frontclip or backclip erase part of an object, it is not permanently erased. Simply dragging the slider bar in the other direction will bring

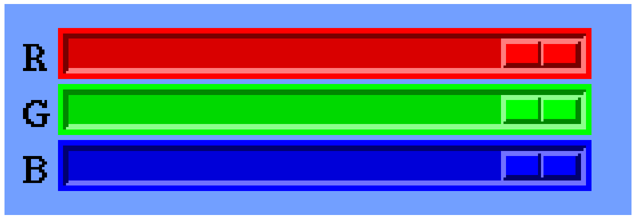



***Figure 1−5.  Background color controls***

the object back into view.

**Vector scale**

The "Vector Scale" slider bar controls the relative length of the current vectors, (see Fig. 1−4).  This adjustment allows you to scale the length of the vectors to your own liking.  Although the length of the vectors may increase when the slider is dragged to the right, the length is always proportional to the magnitude of the data from which the vectors were drawn in the first place.

**Background color controls**

The background color of the main GMV window can be changed by using the background color slider bars located in the top center of the window, (see Fig. 1−5).  Drag the controls back and forth

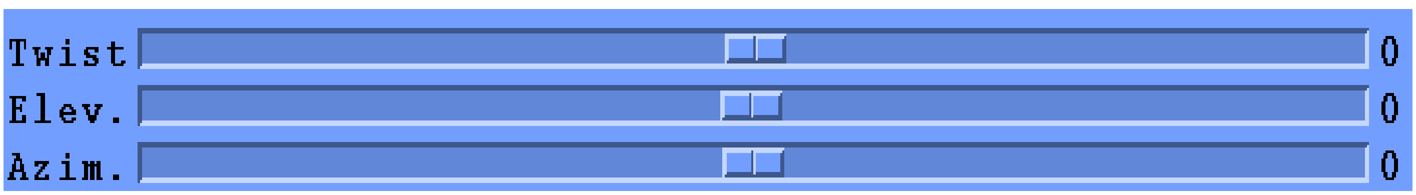



***Figure 1−6.  Twist, elevation, & azimuth controls***

until the desired combination of red, green, and blue is achieved.

**Twist, elevation, and azimuth**

These three slider bars are located above the main viewer and control the viewing angle, (see Fig. 1−6).  "Azimuth" is the angle on the X−Y plane measured from the X−axis.  It has the same effect as using the left mouse button and moving left and right.  "Elevation" is the angle in the direction of the Z−axis measured from the X−Y plane.  It has the same effect as using the left mouse button and moving up and down.  The "Twist" adjustment cannot be done with the mouse.  The twist slider rotates the object about the X−axis.

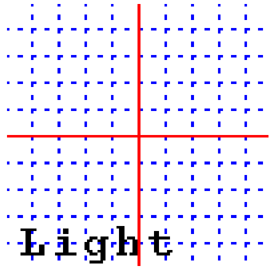



*Figure 1−7.  Light source box*

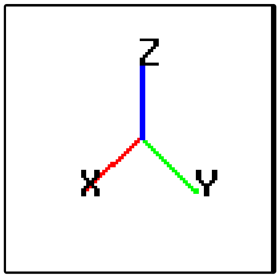



*Figure 1−8.  Axes view box*

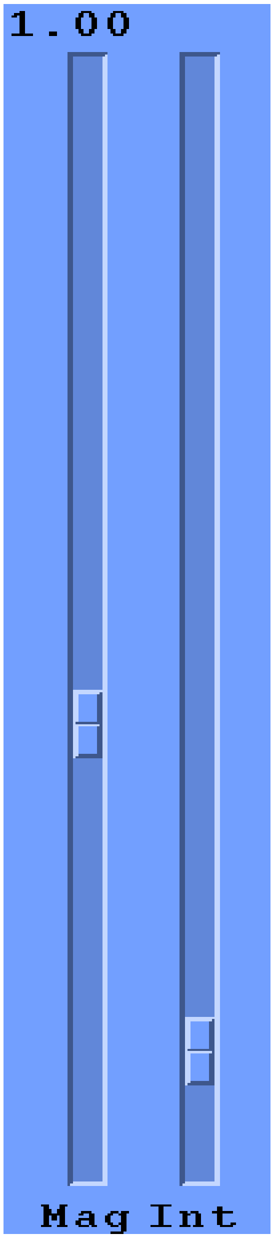



*Figure 1−9.  Magnification and interactivity controls*

**Light source box**

The light source box is used to control the light source.  The box is located near the upper right corner of the window and is labeled "light," (see Fig. 1−7).  By default, the light source is located at the center, but this position can be changed by clicking the mouse anywhere in the light box.  The crosshairs will move to the place where the mouse was clicked.  The new location of the light source is where the two red lines intersect.  The light source is an infinite light with parallel rays always in front of the screen.

**Axes orientation view box**

This box is located up and to the left of the light source box, (see Fig. 1−8).  The box shows the orientation of the X, Y, and Z axes at all times, even if the axes in the main viewer are turned off.  It is used mainly for reference.

**Magnification and interactivity slider bars**

The magnification slider bar (Fig. 1−9) is labeled "Mag" on the bottom and has the magnification factor displayed at the top (1.00 is default).  Sliding the bar up and down changes the size of the object in the main viewer.  The same effect can be accomplished with the right mouse button, (see **The mouse controls**).

The interactivity slider bar (Fig. 1−9) is labeled "Int" on the bottom.  It controls how much of the object in the main viewer remains visible when it is being manipulated. It is used to speed up the interactive drawing process, especially when the visualized data is very complicated and tends to slow the machine down too much. Move the slider up to display fewer object elements, thus increasing interactivity.

## The File Selecton Menu:

GMV opens with a File Selection Menu (see Fig. 1−10). Use this menu to select a file to process. Near the top of the window is a box labeled "Filter." The filter controls what type of files will be displayed in the "Files" box below. For example, a filter such as "/usr/people/guest/*.inp" would display only files in the directory /usr/people/guest that have the extension ".inp". This is very useful for sorting GMV input files from the other files in that particular directory. Clicking on

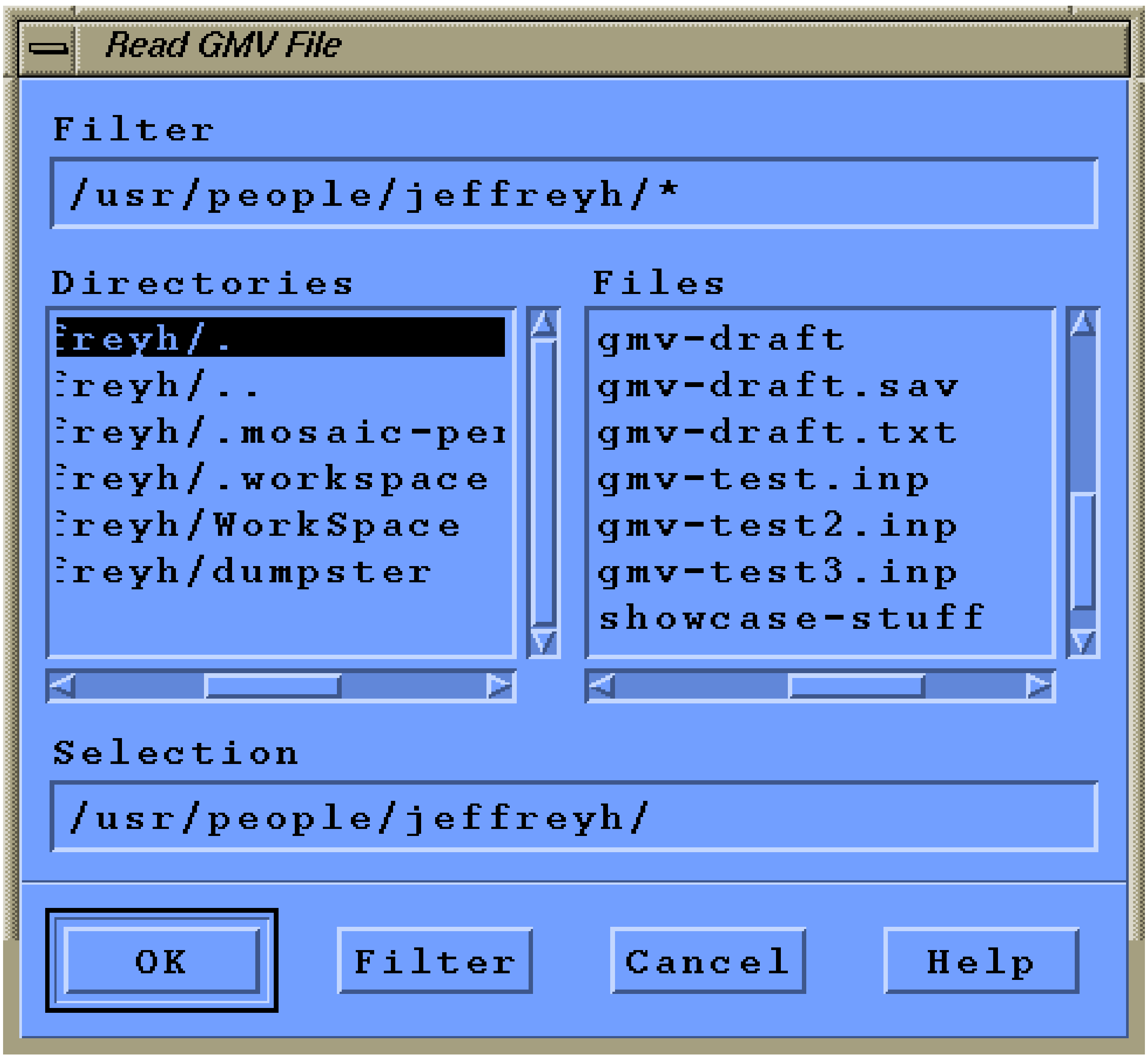



***Figure 1−10. The File Selection menu***

the "Filter" button near the bottom of the window activates the current filter.  Pressing "Enter" on the keyboard while the mouse pointer is in the filter box has the same effect.

Once the directory with the GMV input file has been located, you must choose from the list of files in the "Files" box.  To choose a file, use the scroll bar to position the file name within view and click on the file’s name once.  The name will appear highlighted in the "Files" box and will also be copied down to the selection box near the bottom.  If the exact location of the file is known, the name can be typed into the selection box manually.  Now click on OK to launch the file into GMV.  The previous steps may be skipped if you simply double click on the file name.

If, after the file has been chosen, a watch appears, the selected file is a valid GMV input file and GMV is processing it.  If the file is not a valid input file, a message box will appear stating this, and another opportunity will be given to choose a file.

# *The File Menu*

## Read GMV file:

The first option in the "File" menu is "Read GMV File."  Choose this option, and a pull down menu will appear.  The pull down menu contain three options: New Simulation, Same Simulation, and Auto Read − Same Simulation.

**New Simulation**

The New Simulation option enables GMV to read a file that was generated by a different simulation than the simulation that created the current GMV input file.  Selecting this option will display the File Selection menu (see Fig. 1−10) to select the next input file.  Then the current custom menus will be destroyed while new custom menus for the data on this input file will be created.  The first image will display either nodes, cells, or polygons following the rules used when GMV is started (see Chapter 1).  Only the view angles, magnification, and material colors will be the same as the last image from the previous GMV file.  The 3−D plot box, the subset ranges, and the field data ranges will all be reset to reflect the data in the new GMV file.

**Same Simulation**

The Same Simulation option allows GMV to read a file that was generated as a different time step from the same simulation as the simulation that created the current GMV input file but with a different cell configuration.  Selecting this option will display the File Selection menu (see Fig. 1−10) to select the next input file.  The current custom menus are not destroyed .  The image displayed after reading the input file will contain exactly the same attributes as the image from the previous file.

The new image will be displayed much faster after the file is read since the custom menus do not have to be recreated.  Also, any cutlines, cutplanes, isosurfaces, and isovolumes that existed in the previous image will be automatically calculated and displayed.  The 3−D plot box, the subset ranges, and the field data ranges remain the same for successive implementations of the Same Simulation read option unless manually reset, an attributes file is read, or until a file is read with the New Simulation read option.

**Same Simulation, Same Cells**

The Same Simulation, Same Cells option is similar to the Same Simulation option, except that the cell configuration must be the same as the current GMV file. In other words, the cells must contain the same node numbers. A new cell face list and cell edge list will not be recalculated.

The new image will be displayed much faster after the file is read since the custom menus do not have to be recreated and cell faces and edges remain the same.

**Auto Read − Same Simulation**

The Auto Read − Same Simulation option allows GMV to automatically read a time series family of input files created from the same simulation but with possibly different cell configurations. The filenames to be read by this option must all be the same except for a numeric suffix. The numeric suffix must all be either a 3 or 4 digit number within the family. Also the current GMV file must be a member of the family.

The files are read in a user determined sequence and the image produced after a file is read will have the same attributes as the last image from the previous file. As in the read Same Simulation option, it is important to have the plotbox, field data range and subsets set to values that reflect data for the family of files. The attributes can be changed, however, by pausing the sequence and manually setting attributes or reading an attributes file. You can then resume the sequence or start over.

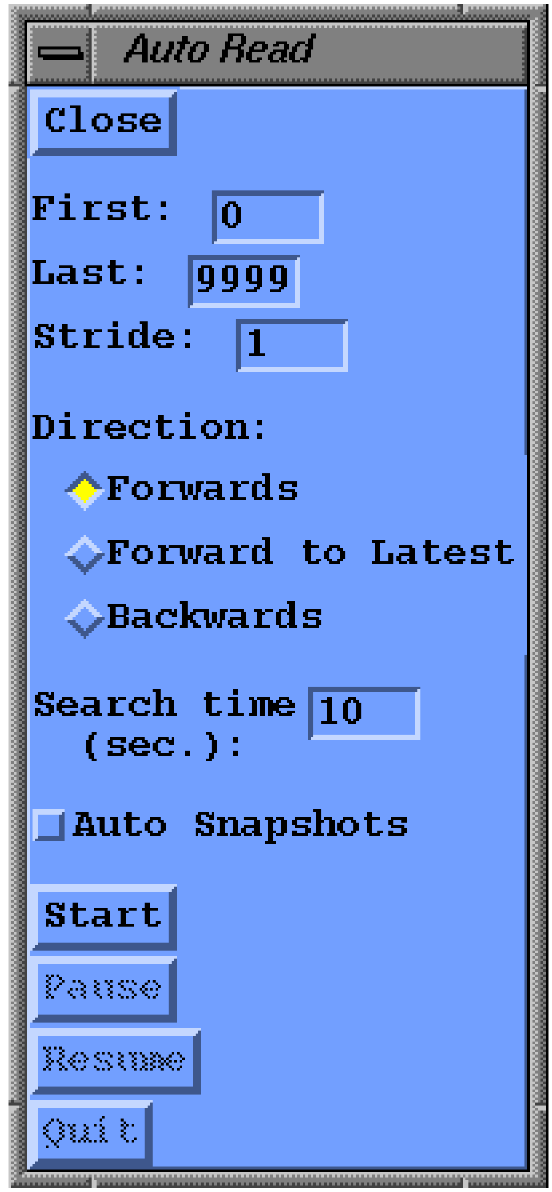



***Figure 2−1. Auto Read Menu***

Selecting this option will display the Auto Read menu (see Fig. 2−1), unless the current file does not contain a numeric suffix. In the "First": and "Last:" text boxes, enter the range of input files to read. Enter a stride (skip value) in the "Stride:" text box. Next, select one of the direction options.

The "Forwards" direction option reads files form first to last incremented by the stride. The "Forward to Latest" option looks for the latest existing file within the specified range. This option is useful to view the latest complete GMV file as the files are being generated by a simulation code. The "Backwards" option reads files from last to first decremented by the stride.

In the "Search time (sec):" text box, enter the time interval GMV will use to search for the next file in the sequence, not including the time to read a file. To sweep through a series of files as fast as possible use a 1 second interval  If you would like snapshots of the first image displayed after the next file is read, press the "Auto  Snapshots" button to display the control menu.  See below for the description of the Auto Snapshot menu.

Press the "Start" button to start reading files.  Press the "Pause" button to pause the file search.  Use "Pause" when you want to closely inspect the current image or when you want to change the current image.  Press the "Resume" button

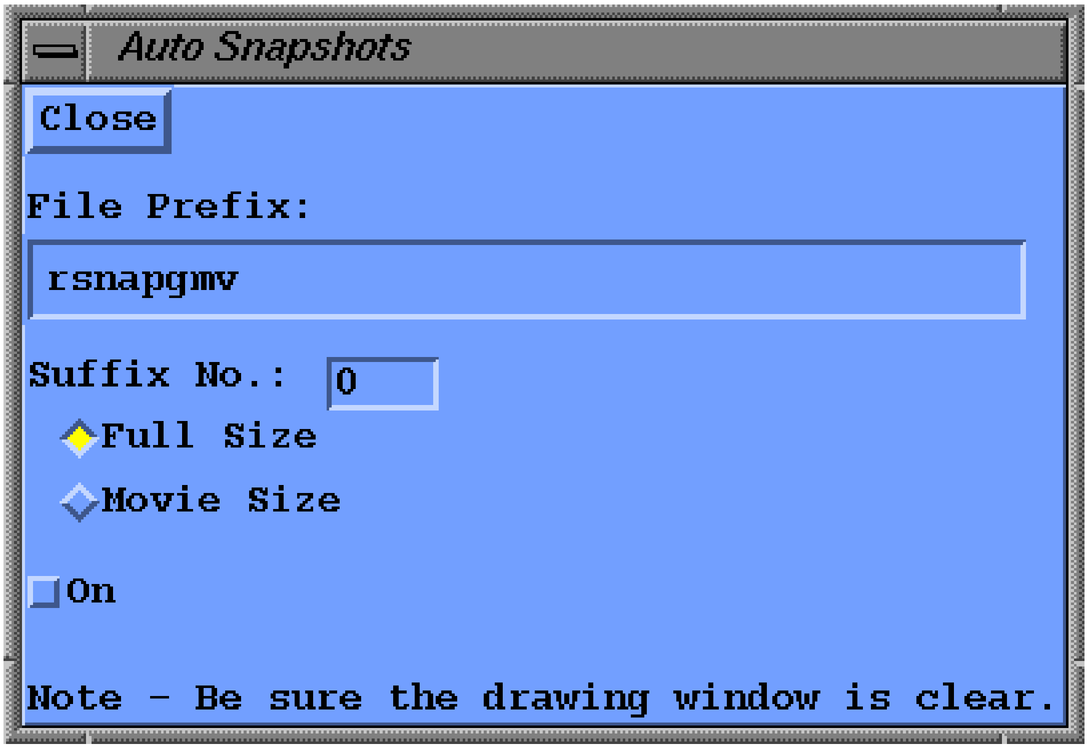



***Figure 2−2.  Auto Snapshots Menu***

to continue the file search after a pause.  The "Quit" button stops the file search. Note that if you change the image during the search, subsequent images will use the image attributes of the image you changed.

To automatically  create a snapshot from each file, press the "Auto Snapshots" button and the Auto Snapshots menu appears (see Fig. 2.2).  Enter the file prefix for the rgb files in the "File Prefix" text area.  Enter the start of the 4 digit suffix number sequence in "Suffix No.:" and select either "Full Size" or the 640x512 pixel "Movie Size" for the saved images.  Select the "On" button to create snapshots when starting or resuming Auto Read.  Be sure that the drawing window is not obstructed during Auto Snapshots.

**Auto Read − Same Simulation, Same Cells**

The Auto Read −  Same Simulation, Same Cells option except that the cell configuration must be the same as the current GMV file.  New cell face lists and cell edge lists will not be recalculated.


*Figure 2−3.  Prompt for attribute file name*

## Put and get attributes:

Attributes are the collective sum of all the options available in GMV, (see Fig. 2−3).  Normally when GMV is first run, all of GMV’s options are set to their defaults.  For example, the "Twist," "Elevation," and "Azimuth" slider bars are all set to zero and the magnification factor is set to one.  However, suppose you have worked on viewing the object from a certain angle and have created isosurfaces and a cutplane, or you want to apply the attributes to a different time step of the same simulation.  If you want these to reappear the next time you start GMV,  you can save the attributes in a file and retrieve them later.  Choosing the "Put Attributes" option prompts you for a name under which to save the attributes. After a name has been chosen, click OK to save the file.  The "Put Attributes" function is necessary in order to create time sequence movies of a simulation (See **The GMV Movie utility**).  The attribute file may be retrieved later by invoking the "Get Attributes" option under the "File" menu.  Take note that when a set of attributes is saved and then immediately retrieved, the name of the attribute file will not show up in the "Get Attributes" list of files until the "Filter" button is clicked, updating the file list.

## Snapshot:

Snapshot is a tool that can create image files of the currently displayed GMV data, (see Fig. 2−4).  A GMV snapshot can only be two sizes: full size and movie size.  Movie size (640 x 512) is the smaller one and can be used to create a sequence of images for animation.  Snapshot images are always saved in the SGI−RGB format.  There are plenty of freeware utilities available to convert to other formats.

After the "Snapshot" option has been chosen, a small window will appear.  In the first box you must enter a name for the snapshot.  Underneath the file name, the "Full Size" and "Movie Size" options appear.  Choose one.  The radio button next to the currently selected snapshot size will be yellow.  When all is ready, click on "Snap."  After the image is stored, click on the "Close" button and continue

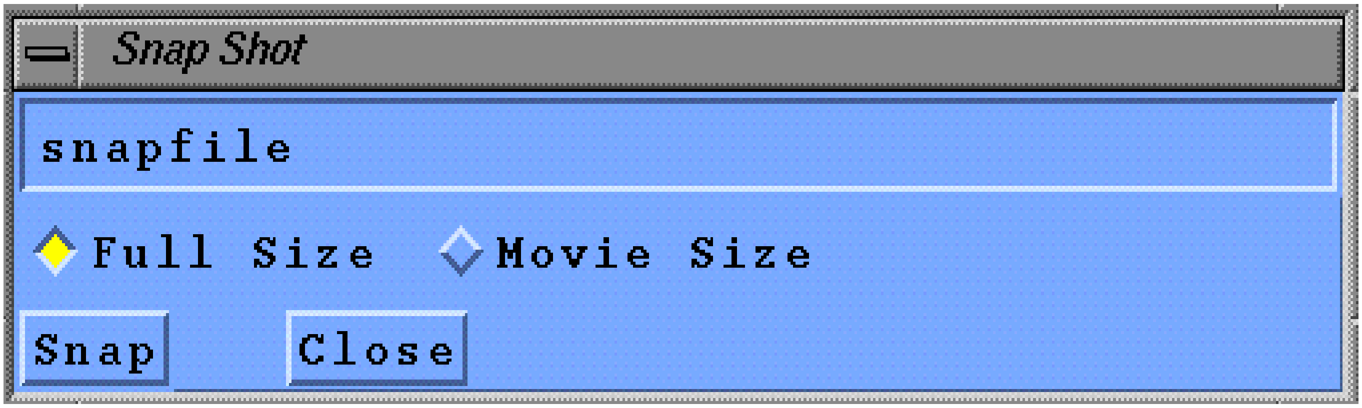



***Figure 2−4.  SnapShot menu***

work.  The snapshot File will be saved in the current directory.

## Quit:

Choose this option to quit GMV.

# The Display Menu

## Nodes:

Nodes are points in three−dimensional space with X, Y, and Z coordinates. They may also contain various types of data such as material or velocity information.

**Viewing nodes, their vectors, and numbers**

When the "Nodes" option is chosen from the "Display" menu, a nodes menu like Fig. 3−1 will appear. From this menu, you may choose which aspects of the node data to view. Click the "Apply" button to activate requested selections from the menu. Below the "Apply" button in the menu are three options that may be chosen individually or simultaneously. Clicking on the "Nodes" button will cause the nodes to appear as colored dots on the screen, where the colors are determined by the "Color By:" options.

Below the "Nodes" button is the "Numbers" selection box. When this option is activated, each node's respective number will be displayed next to its point on the screen.

Below the "Numbers" button is the "Color By" section. In this section you choose whether you want to color the nodes by materials, a node field or a flag.

By default, the node data is colored according to each node's material number. However, GMV incorporates provisions to display the node data as a blue to red color intensity color bar according to or any user−defined field. The node can also be colored by a flag type value. To color the nodes by a the current field value, click on the "Node Field:" button. To select a different field, click on the "New Field" button to pop up the Node Field Selection menu (see Fig. 3.2). To color the node by the current flag type, click on the "Flag" button. To change the current flag


*Figure 3−1. Nodes menu*

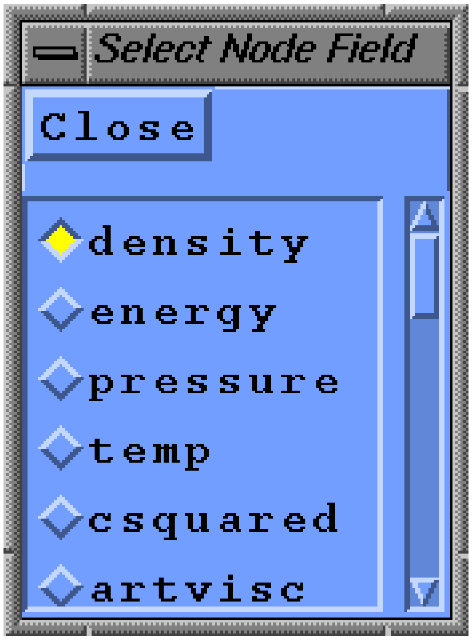



***Figure 3−2.  Node Field Selection Menu***

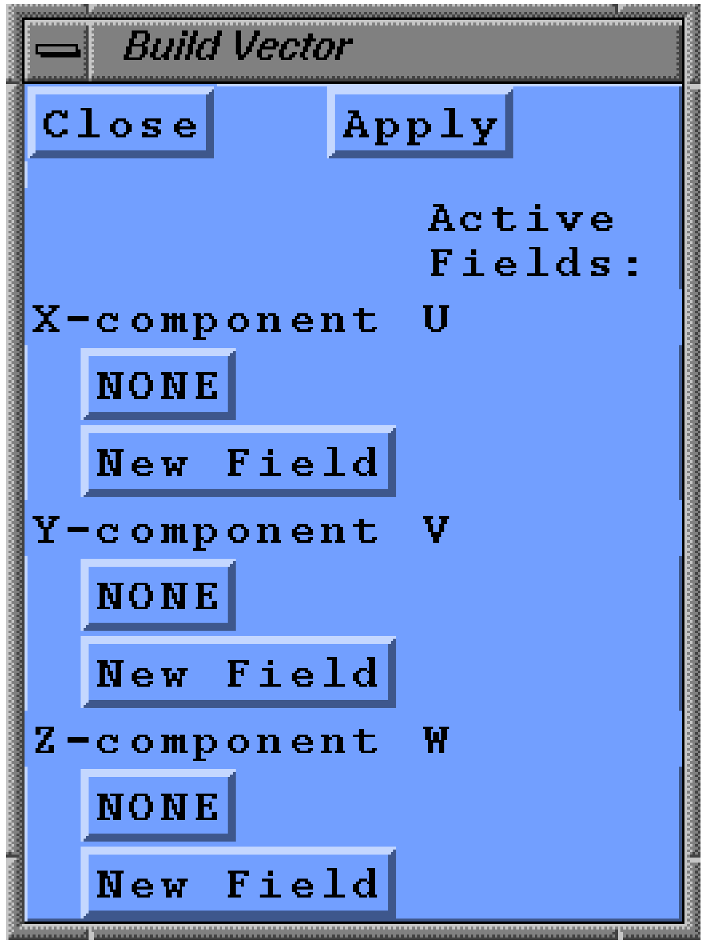



***Figure 3−3.  Build Vector submenu***

type, click on the "New Flag" button and select a flag type from the Flag Selection menu.  Only one of material, node fields, or flags may be selected at a time.  When a node field is selected to color the nodes, a color bar scale will appear on the far left of the main viewer that  gives you the range of values in relation to the color bar.

To see any nodes that have a 0 material number, press the "Show nodes with 0 material nos." button.  Any nodes with 0 material number will have the text color as a material color.

**Vectors:**

Choose the "Vectors" button to build and draw vectors.  To turn on the vectors, click on the button.  A submenu will appear.  The menu contains a toggle button to toggle the vectors on and off.  When this button is selected, each node’s vector data will be displayed according to the current X, Y, and Z components of the vector.  The vector is colored by the node material number with a  white or black tip, depending on the background color.

There is also the "Build Vector" option.  Click here and a submenu similar to Fig. 3−3 will appear.  With this option, you can tailor the X, Y, and/or Z components of the vectors to any field; default is the velocity data (U, V, and W).

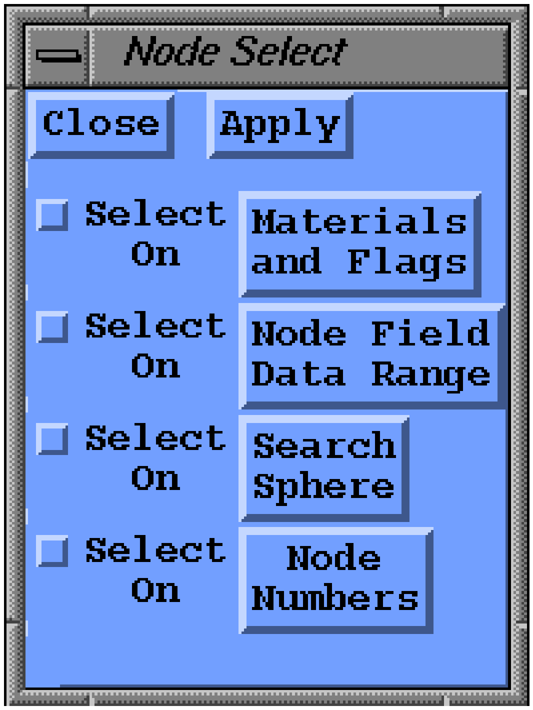



***Figure 3−4.  Node Select submenu***

To change a vector component, click the "New Field" button under the vector component to change.  The Node Field Selection menu is then displayed (see Fig. 3−2).  Choose the desired field for that vector component from the list of fields.  A vector component's current field will be displayed to the right of its name under the heading "Active Fields".  Choose "NONE" to zero out a vector component.

**Selecting nodes to display**

If you want to select certain nodes to display, click on the "Select" button in the "Nodes" menu and the "Node Select" menu is displayed (see Fig. 3−4).   Use this menu to select nodes by materials and/or flags, by a data range of a node field, by a search sphere, or by node numbers.  To specify a selection type, click on one of the category buttons to bring up a selection menu.  After specifying the selection criteria, click on the "Select On" toggle button to use the selection type.  If more than one selection type is set on, GMV will use a logical and to combine the selections.  Click on the "Apply " button to process the search criteria.

**Selecting nodes by Materials and Flags**

To select nodes by materials and flags, click on the "Materials and Flags"  button and the Node Materials and Flags Menu appears (see Fig. 3−5).  The available materials will be listed on the left side of the menu.  Following the materials are all the flag types,if any, followed by the different flag values.  Between each column of selection criteria are the two logical operators "AND" and "OR."

Highlight the desired materials, then highlight the desired flag values.  The "On" and "Off" buttons are used to select or unselect all of the materials or

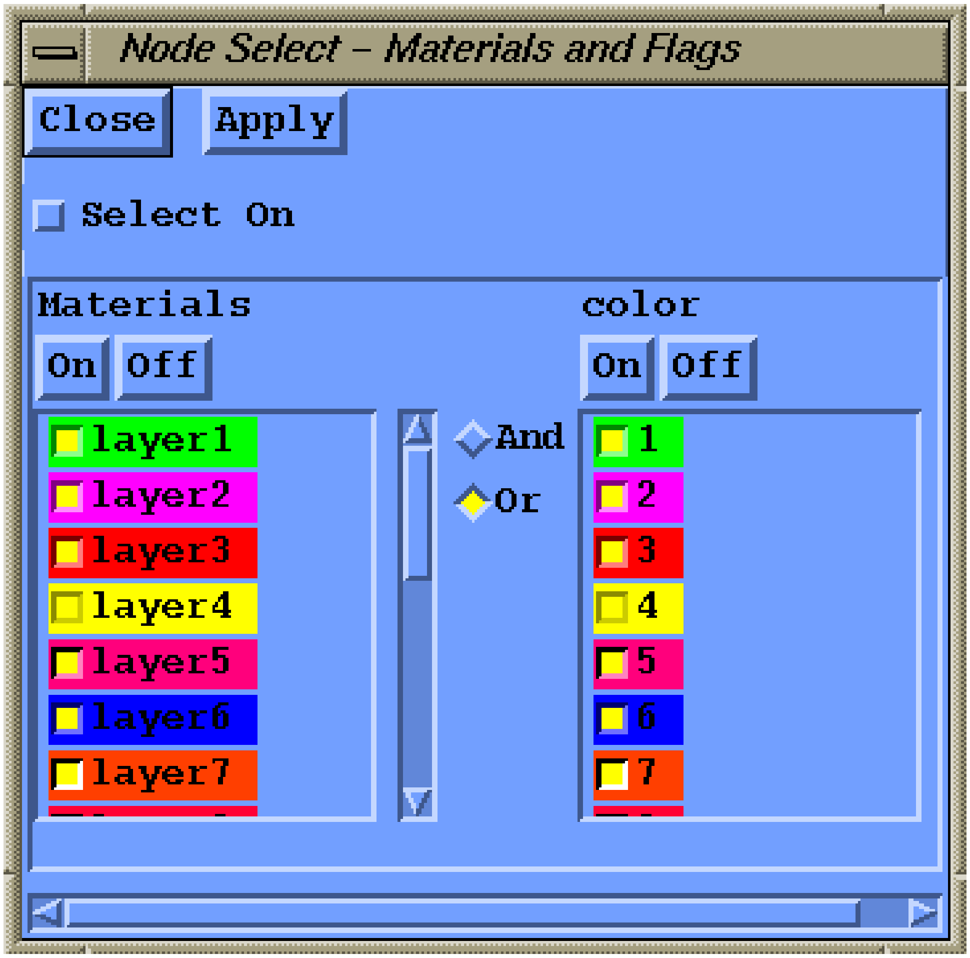



***Figure 3–5. Node Materials and Flags submenu***

flags. Next, decide on the logical operator (and/or) to build a left–to–right boolean operation.

**Selecting nodes by Node Field Data Range**

To select nodes by a node field data range, click on the Node Field Data Range button and the "Node Field Data Range" menu appears (see Fig 3–6). Select a field to operate on by clicking the "New Field" button to pop up the Node Field Selection menu. The minimum and maximum values of the field will then be displayed. Then enter your minimum and maximum data range for node selection in the text fields. Click on the "Reset" button to reset the field minimum and maximum values in the text field.

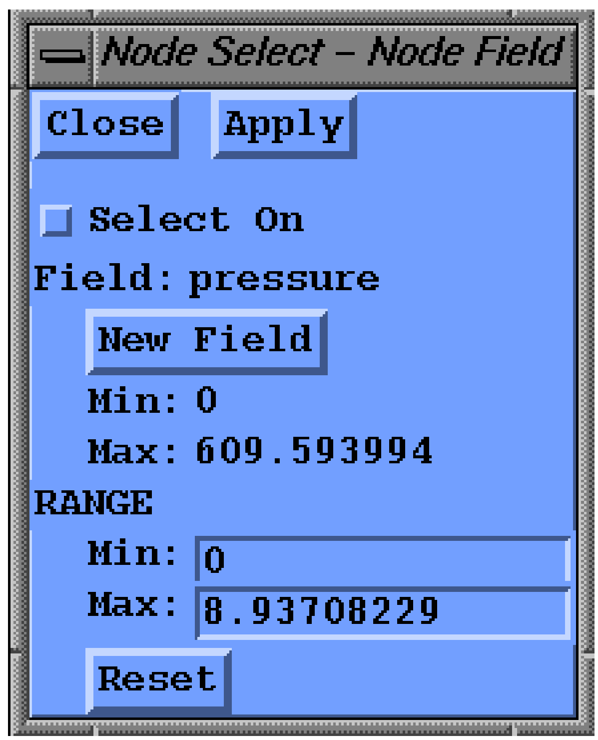



***Figure 3–6. Node Field Data Range submenu***

**Selecting nodes by Search Sphere**

To select nodes within a user defined search sphere, click on the "Search Sphere"

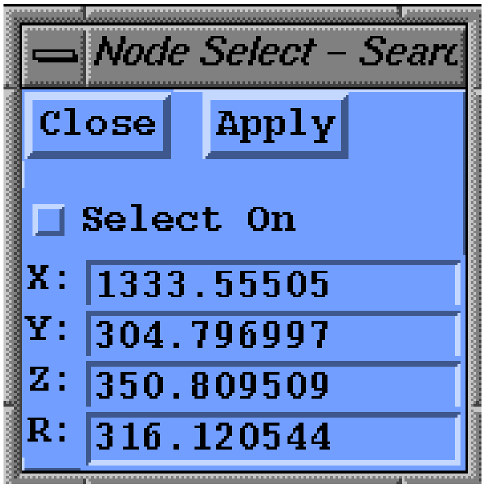



*Figure 3−7. Node Search Sphere submenu*

button and the "Node Search Sphere" menu appears. To define the search sphere, enter the x, y, and z coordinates of the center of the sphere and the sphere's radius.

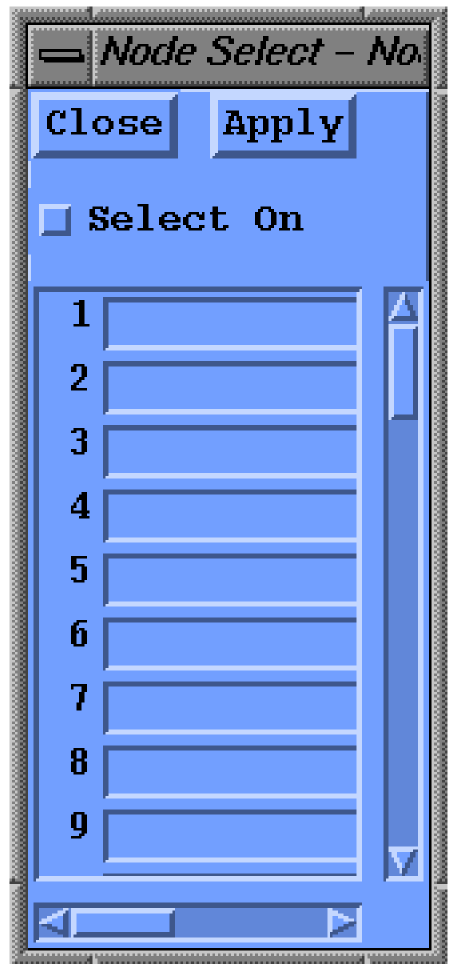



*Figure 3−8. Node Number submenu*

**Selecting nodes by Number(s)**

To select nodes by numbers or a range of numbers, click on the "Node Numbers" button and the "Node Numbers" menu pops up (see Fig 3−8). The menu contains 50 lines where individual node numbers, a range of node numbers, or a range of node numbers with a stride can be entered (one entry per line). A colon (:) is used as the delimiter when defining a range of numbers or a range of numbers with a stride. The format used to define a range of node numbers is first:last, eg. 1:10. The format used to define a range of node numbers with a stride is first:last:stride, eg. 20:100:10. All node numbers must be greater than 0, and any number greater than the maximum node number will be reset to the maximum node number. If a line contains an invalid character, an error message will appear in the Node Select menu indicating the line number with the error.

**Activating node selection**

Be sure to click the "Select On" toggle button in the Node Select menu or on the specific selection menu. Then click on the "Apply" button to start the selection process.

## Cells:

Cells are three−dimensional fixed shapes in space, such as cubes, pyramids, or prisms. Cells are defined by the nodes at their vertices in the input file.

**Viewing cell faces, edges, and numbers**

Choosing the "Cells" option under the "Display" menu brings up the "Cells" menu, (Fig. 3−9). Here you may choose which aspects of the cells you wish to view. The "Faces" option displays the face of the cells as an interpolated blended color polygon where the colors are based on the selected cell or node field. If the cell face is colored by a node field, the "Contours" option under "Faces" displays contour lines on the faces. The contour lines are drawn at the intervals shown on the cells Color Bar. The "Shaded" option under "Faces" turns on the lighting model and shades the faces. The "Edges" option displays the edges of the cells as a colored wireframe image. The edges are colored by material or flag colors. If both "Faces" and "Edges" are selected, the edges are colored grey. The "Numbers" option draws the cell number at the center of the cell in the edge color for the cell. Cell faces, edges, and numbers may be displayed simultaneously. Simply click on the box next to the desired option just as is done in the "Nodes" menu. Click on the "Apply" button to activate the selections.

**Coloring cells by materials, fields, and flags**

Cell edges are colored by material color, flag values, or a blue−to−red intensity color for cell centered or node centered field values. If node field values are selected the edges are smoothly colored. The cell faces can be colored by material or flag values as well as a blue−to−red intensity color of cell centered or node centered field values. If node field values are selected, the cell faces are smoothly colored.

To see cells that have a 0 material


*Figure 3−9. Cells menu*

number, press the "Show cells with 0 material no." button.  Any cells with 0 material number will have the text color as a material color.

### Cell vectors

If there is cell−centered data, cell vectors can be built.  Click on the "Vectors" button to bring up the appropriate submenu.  The procedure to display and build vectors is the same as node vectors.

### Selecting cells to display

This is done in the same manner as described in the previous section about nodes and their materials and flags, cell field range, search sphere  and cell number(s).  Additionally, cells can also be selected by a node field range for the nodes that define the cells.  The "Cell Node Field Range" menu contains two additional buttons, the "Any" and the "All" toggle buttons.  The "Any" button will select a cell if any of its nodes fall between the node range value.  If you want to select  cells with all nodes falling between the node field range, click on the "All" button.

### Explode

Explode allows you to separate groups of cells based on material or flag data.   Clicking on Explode pops up the Cell Explode submenu, (see Fig. 3−10).  Use the "Explode %" slider to adjust the amount of separation of cell groups as a percentage of the distance.  Use the "Explode On" radio buttons to select the material number ("Mat. No") or flag type to group the cells.  Click on "Apply" to display the separated groups of cells.  To return to the initial setup, return the slider bar back to zero and press apply.

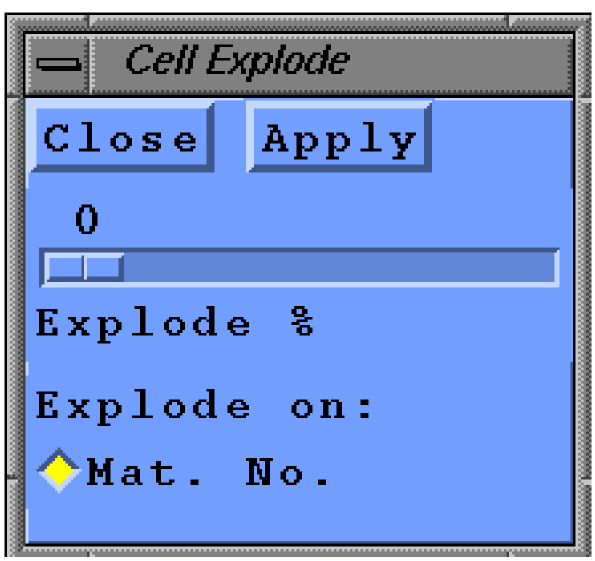



***Figure 3−10.  Cell Explode submenu***

## Polygons:

The "Polygons" menu is used to view the surfaces made by the input polygons, (see Fig. 3−11). Polygons are surface facets that are shaded according to the location of the light source. The polygons are colored by material color as specified in the input file.

**Shading and outlining polygons**

To change the way GMV displays polygon data, choose the "Polygons" option from the display menu. The "Apply" button is used to activate the desired selection from this menu. The polygons can be outlined, shaded, or both, depending on your preference. The way the polygons are "Shaded" depends on the location of the light source. This location of the light source can be changed by using the light source box in the upper right corner of the main GMV window. See the information on page 1−7. When the "Lines" option is selected, GMV draws lines between the vertices of the polygon, to create a wireframe image. If both "Shaded" and "Lines" options are selected, the polygon lines are colored white.

**Selecting materials to display**

The surface materials may be turned on or off. The materials are listed at the bottom of the "Polygons" menu. For convenience, there is an on and off button to turn all the materials on or off at once. Use the material toggle buttons to select individual material surfaces for display. Press the "Apply" button to activate the selection.

**Changing explode percentage**

Use the "Explode%" slider to separate surfaces by material, giving an exploded view of the surfaces.


***Figure 3−11 Polygons menu***

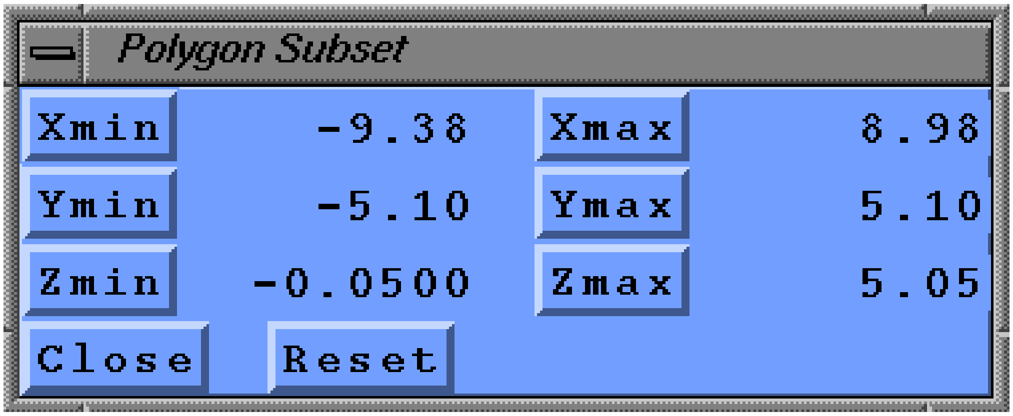



***Figure 3−12.  Polygon subset submenu***

**Selecting a polygon subset**

When you click on the button labeled "Subset" in the "Polygons" menu, a submenu labeled "Polygon Subset" will appear, (see Fig. 3−12).  There are six buttons in the window.  The buttons are used for defining a region of space.  The minimum and maximum values listed next to the six buttons define a cube in space.  Any polygon that does not lie completely within this region of space will no longer be shown in the main viewer. To change a minimum or maximum coordinate value, first position the view angles as multiples of 90 degrees.  This is necessary because the limiting plane value is determined by its intersection with the axis selected.  Next click on the box next to the value you want to change. Now move the mouse into the main viewer.  The mouse pointer will no longer be an arrow, but two crosshairs similar to a plus sign.  Click the mouse when the crosshairs are located at a point in space with the desired coordinate to be used as a maximum or minimum value.  The new limiting value will be displayed next to the appropriate button in the "Polygon Subset" window.  Additionally, any polygon not falling within the new region of space will be erased.
(Here is a helpful hint: position the axes so that the axis with the coordinate sought after is parallel to the screen.  This will allow you to more accurately select a value.)  Clicking on the close button will close the subset window, leaving any changes in effect.

**Changing material order**

The material order function comes into play when two or more polygons occupy the same region of space.  Because only one of the stacked polygons may be shown at a time, you must decide the order of precedence for the materials so that GMV will know which polygon to draw.  Material order is also important when more than one material is transparent.  Transparent materials must be in a back to front order in order to be drawn correctly.  By default, material


***Figure 3−13. Material Order submenu***

number 1 is first, followed by material number 2 and so on.

To change the current material order, click on the button labeled "Mat Order" in the "Polygons" menu. A submenu similar to Fig. 3−13 will appear. On the far right you will see the current materials listed in order. To change the order, click on each material button from the left−hand column in the desired order. The materials will be listed in the middle column in the proposed order. Each material may be chosen only once. If you make a mistake in the material ordering process, click on "Undo." This will clear the order listing of materials and allow you to start over. When the materials are in the desired order, click on "Apply" and the changes will take effect. These changes and new order number will appear on the right−hand column. Click on "Close" when finished.

## Tracers:

Tracers are points in space used for monitoring data in locations where nodes do not exist. Tracers are defined by their X, Y, and Z coordinates and by the data they are assigned by the input file. Tracers can be assigned multiple fields of data, such as in Fig. 3−14 where it reads pressure, temperature, and pH data from the input file.

**Methods of displaying tracers**

Tracers can be represented in three different ways by GMV, either as big points, numbers, or values. When the "Tracers" menu is brought up from the "Display" menu bar, you will notice four selection boxes under the heading "Draw as." When the "None" option is chosen, no tracer data will be shown. The "Big Points" option displays tracer data as large, bright icosahedra centered on the tracer location. When "Numbers" is selected, the sequential number of each tracer is displayed. Do not get the tracer numbers confused with the node or cell numbers. When all three are displayed simultaneously, it is hard to tell them apart. Finally, there is the "Values" option. This one tells GMV to display the value of each tracer in the correct location on the screen. Depending on which field is currently selected, the colors for tracer display are a blue to red intensity color depicting the values of the field selected for display. Click on the "Apply" button to activate your selections.


***Figure 3−14. Tracers menu***

**Selecting data field for tracer to represent**

A main color bar pops up inside the GMV main viewer when tracers are displayed. The tracer always takes on the color of its value in the currently selected field. The available fields are listed under the heading "Fields" at the bottom of the tracer window. To select a field, click on the radio button next to its name. The colors of the tracers in the main viewer will change. If the input file read in by GMV contained no tracer data, then there will be no fields to choose from in the "Tracers" menu. Selecting the "Close" button will close the menu, leaving any changes you have made intact.

**Selecting tracers to display**

Tracers can be selected for display by number(s). Click the "Select" button to display the "Tracer Select Menu" (see Fig. 3−15). As in selecting nodes or cells by number, there are 50 lines where a tracer number, a range of tracer numbers, or a range of tracer numbers with a stride can be entered. Use a colon (:) as the delimiter for a range of numbers and for a range of numbers and a stride. For example to select tracers between 1 and 10, enter 1:10, to select every tenth tracer between 20 and 100, enter 20:100:10.

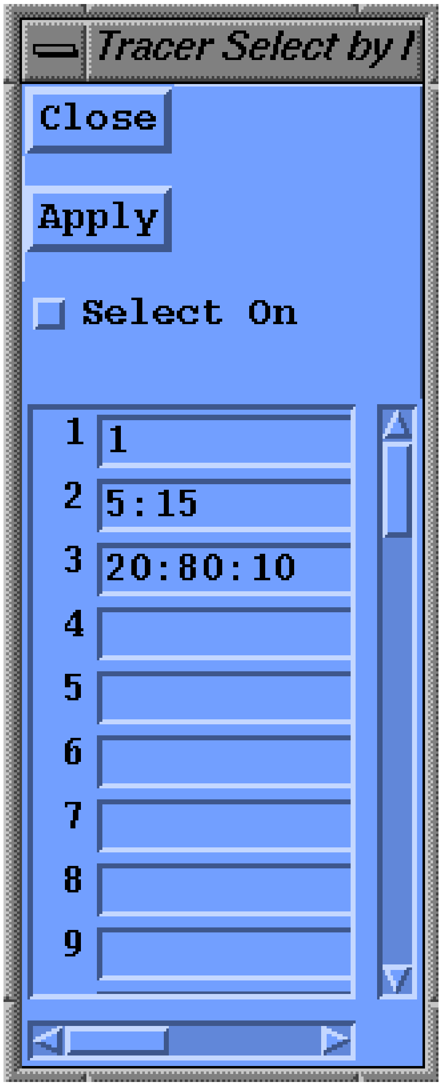



*Figure 3−15. Tracer Select submenu*

**Display tracer history**

Tracer histories are displayed as a set of colored line segments connecting with the big points, numbers or values display of the current tracers. The line segments connecting the history locations are smoothly colored according to the value of the selected field for up to 250 history points per tracer. Click the "History" button to turn on the tracers history display option and click the "Apply" button to initiate the option.

Trace histories are automatically read from previous GMV input files when the trace history option is applied. In order to read trace histories, all of the GMV input files must have the same file name prefix and must have a 3 or 4 digit number as the file name suffix. GMV will only read history files that exist in the directory of the current GMV file and will continue to read the files until there are no more files to be read or until 250 files have been read. Once a set of history files are read, GMV will not read any more trace histories until another GMV input file is read using the Read GMV option of the File menu.

# The Calculate Menu

## Cutlines:

The "Cutlines" function in the "Calculate" menu allows up to twenty cutlines to be generated. A cutline is the centerline of a cylinder with a user defined radius. The field value of any node that lies within the cylinder is projected onto the cutline and displayed as a blue−to−red color coded line.

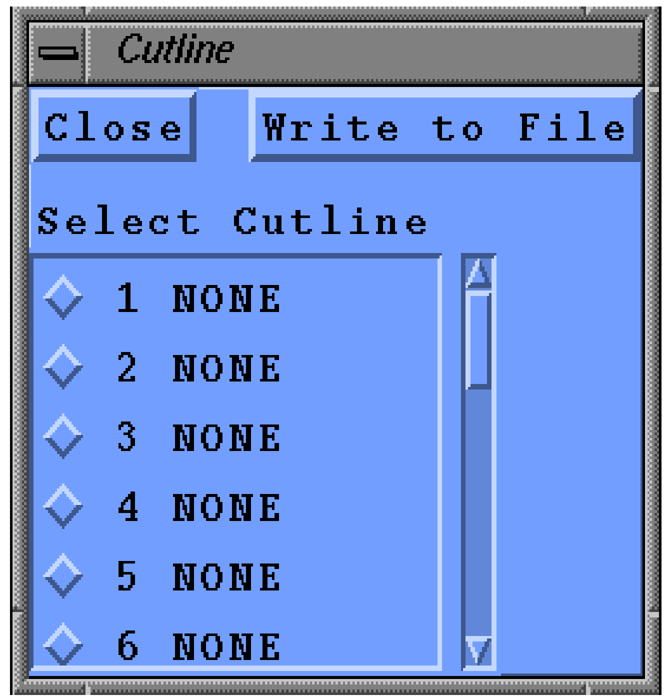



*Figure 4−1. Cutline Selection menu*

**Selecting a cutline**

Choose the "Cutlines" option from the "Calculate" main menu bar and a menu similar to Fig. 4−1 will appear. The numbers are the cutline number and "NONE" indicates that the cutline has not been generated. A field name indicates that a cutline has been generated for that field. Select one of the cutlines and a menu similar to Fig. 4−2 will appear.

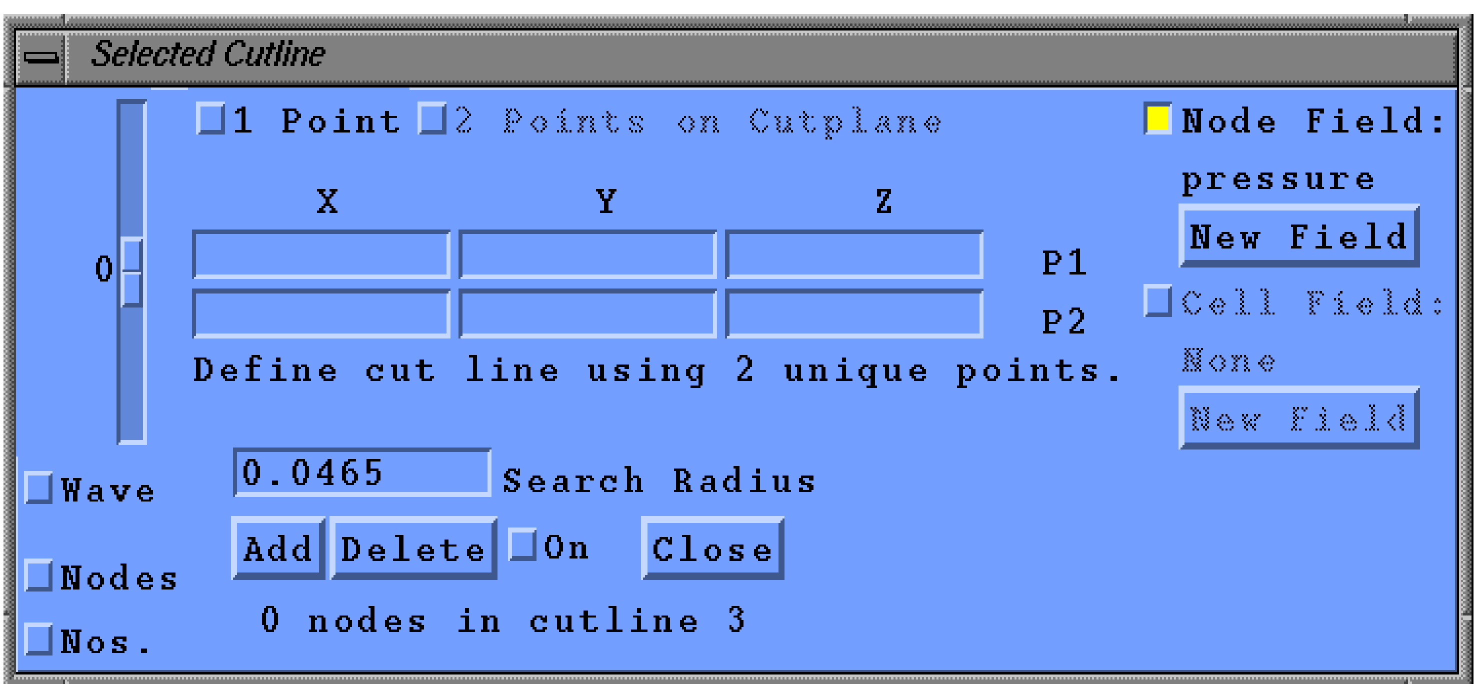



***Figure 4−2. Create cutline menu***

**Creating a cutline**

A cutline can be created in one of three methods. The line can be defined by entering the x, y and z values of the two enpoints of the line. These can be entered on the two rows of boxes identified as "P1" and "P2". A second method is to use the "1 Point" option. Click on the "1 Point" box then move the cursor to the main viewer. A crosshair cursor will appear, place the cursor on a point on the screen and click the left mouse button. A line will be generated normal to the screen with endpoints at the plot box intersections with the line. The third method of defining the line is with the "2 points on Cutplane" option. If a current cutplane exists, select the "2 points on a Cutplane" option and move the cursor to the main viewer. Move the crosshair to a point on the cutplane and click the left mouse button, then move to the second point on the cutplane and click the left mouse button. A line will be defined where lines normal to the screen from the selected points intersect the cutline.

It is possible to select the "2 points on a Cutplane" option when multiple cutplanes are defined. To avoid ambiguity, all cutplanes other than the desired one should be disabled prior to creating the cutline with this option.

When the cutline has been defined, enter the cylinder radius in the "Search Radius" box if the default radius is not appropriate. Hint: Use the "Distance" option of the "Calculate" menu to determine an appropriate distance. Next, select a field whose node or cell data will be color coded along the line. Finally, click on the "Add" button to create the cutline, a message will then be displayed along the bottom of the menu showing the number of nodes selected for this cutline. The corresponding button in the Cutplane Selection will display the field value selected.

**Cutline display options**

The "On" button toggles the cutline display for the selected cutline on or off. The "Nodes" button, when on, displays the nodes selected for the cutline. The "Nos." button displays the node numbers for the nodes selected for the cutline. The "Wave" toggle button and its slider is used to display the function wave of the data along the cutline. Use the wave slider to increase/decrease the amplitude of the wave. The "Delete" option deletes the current cutline, deleting a cutline causes the corresponding button in the Cutline Select menu to display "NONE".

## Cutplanes:

A cutplane is a plane through the simulation onto which data is interpolated. GMV will display only the parts of the cutplane that intersect with the mesh data in the main viewer. Cutplanes are useful for generating color contour plots of data for detailed analysis.

**Main Cutplanes Menu**

The Main Cutplanes Menu is accessible by choosing the "Cutplane" option from the "Calculate" main menu bar. This menu, shown in Fig. 4−3, contains several buttons that allow the specification of how a field is interpolated by a cutplane. Descending the menu, the first button allows the retrieval of a field value from an arbitrary location on a cutplane. The next set of buttons allow the selection of creating the cutplane based on node or cell data, and the particular field dataset to be used in each case. In addition to the ability to select nodes or cells for the desired field data type to display, this menu allows the modification and update of field variables displayed on existing cutplane(s) (in this case, changes affect all active cutplanes simultaneously). This menu allows the selection of particular cutplanes with the use of the toggle buttons in the lower portion of the menu.

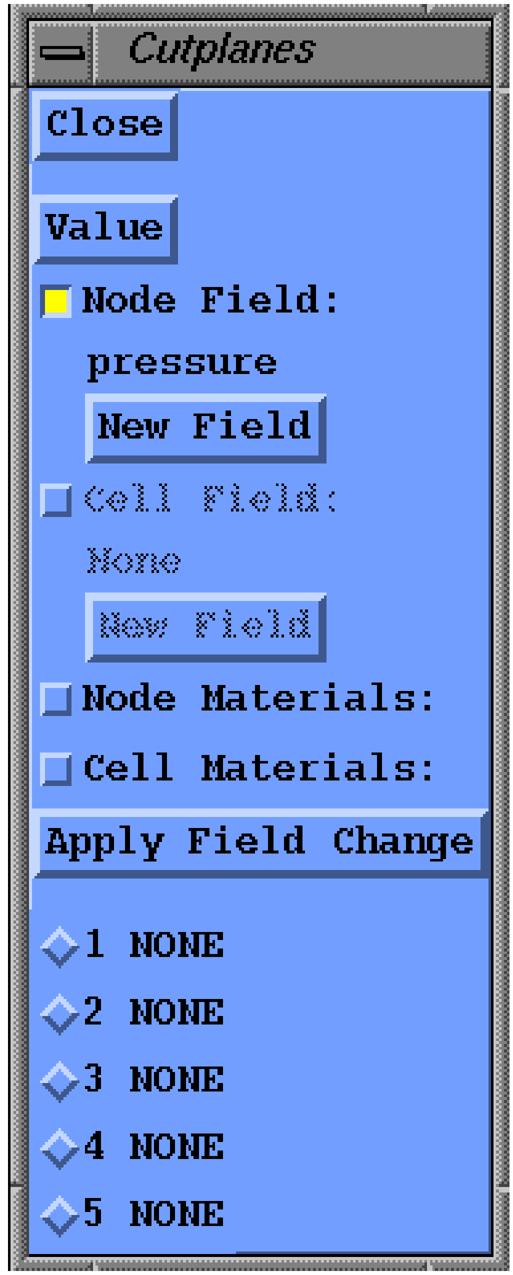



***Figure 4−3. Main Cutplanes menu***

**Value**

The "Value" button is located in the upper left corner of the "Cutplane" menu. The "Value" button can be used to determine the contour value at any point along the cutplane. To determine a contour value, first click on the "Value" button. Move the mouse pointer to the main viewer, at which point it will change into crosshairs. Click the crosshairs anywhere on the cutplane. The value at that point will then be displayed along with the current field name to the right of the "Value" button.

**Node Field, Cell Field or Material**

The "Node Field", "Cell Field" "Node Material" or "Cell Material" toggle buttons select which type of data to be used to color the new cutplane. In the

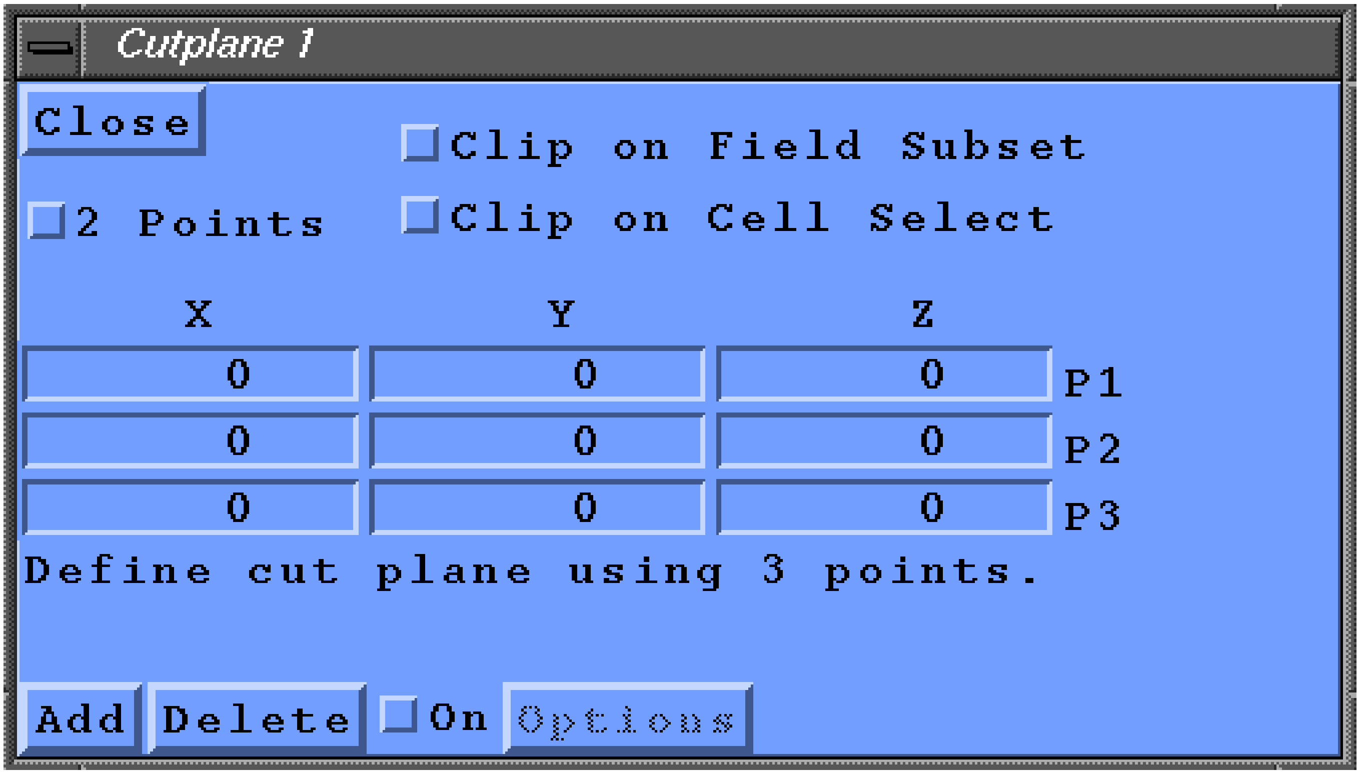



***Figure 4−4. Cutplane menu***

case of the modification of existing cutplanes, these toggles allow changing the current data used to color  the cutplanes.

**New Field**

The "New Field" button under both of the "Node Field" and "Cell Field" selectors allows the specification of the particular data field desired when node or cell data is selected.  This feature may also be used to modify the field used on an existing cutplane.

**Apply Field Change**

The "Apply Field Change" button updates the defined cutplane(s) with any changes made to the field or material configuration.  If multiple cutplanes are defined, field or material changes will be applied simultaneously to all cutplanes.

**Cutplane Selection Buttons**

The last five buttons in the "Main Cutplanes Menu" allow the selection of particular cutplanes.  When a new cutplane is added, the uppermost unused toggle is selected to define the cutplane and specify its location.  The selection of an unused toggle generates a menu similar to Fig. 4−4 for cutplane creation.  When a cutplane has been specified and created, the "NONE" tag in the main menu next to the selection button changes to "ON."  If the cutplane is subsequently toggled off, the "ON" tag changes to "OFF."  If the cutplane is deleted, the tag changes back to

"NONE."  Thus, it is evident which cutplanes are active and how many have been created, in addition to providing the capability to specify a particular cutplane.

**Cutplane Description Menu**

When a particular cutplane from the lower portion of the main menu is selected, the description menu (Fig. 4−4) is created. This menu incorporates the "2 Points" selection button and the data fields where cutplane coordinates may be entered as described above.

**Clip on Field Subset and Cell Selection**

The two "Clip" toggle buttons, found at the top of the "Cutplane Options" submenu, allow you to clip the cutplane according to the subset defined in the subset tool if you choose "Clip on Field Subset", or according to the subset defined by the Boolean expression created using the "Select" option in the "Cells" menu if you choose the "Clip on Cell Select."  When you select one or more of these buttons, the cutplane will only be drawn from data within the limits of the subset, leaving out everything outside the subset.  These buttons must be selected before adding a cutplane in order for them to take effect.

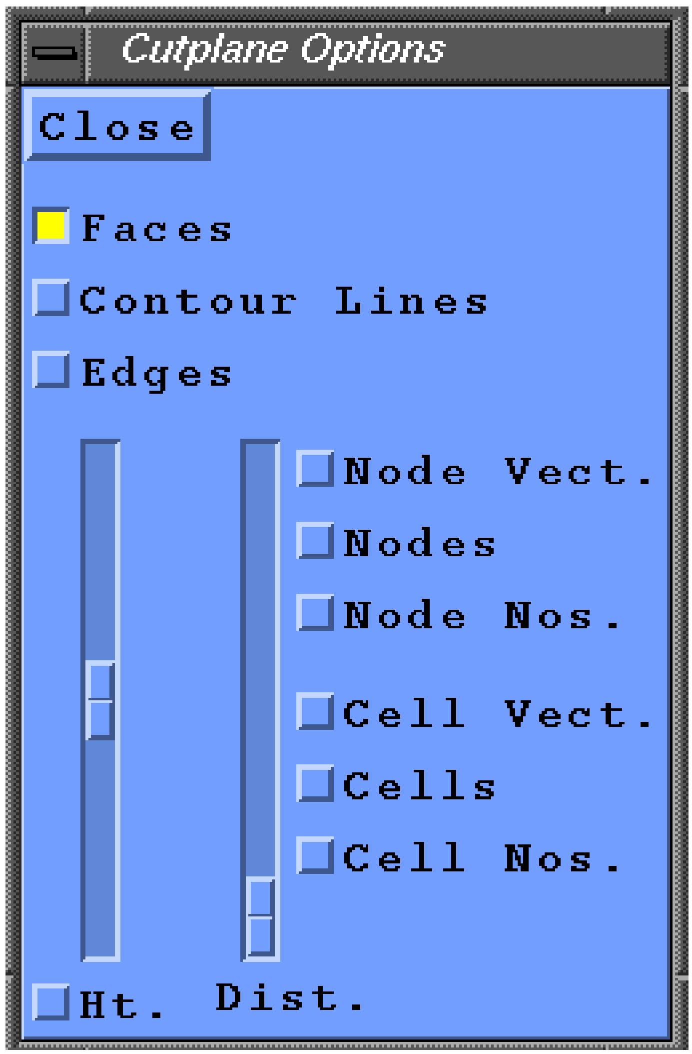



*Figure 4−5.  Cutplane Options menu*

**Cutplane Options**

The button labeled "Options" in the cutplane menu pops up the submenu called "Cutplane Options."  In the cutplane options submenu (Fig. 4−5) there are two slider  bars and several toggle buttons. Cutplane option changes apply only to the currently selected cutplane.

**Faces**

One of the toggle switches in the "Cutplane

Options" submenu is an option labeled "Faces." This toggle controls the visibility of the colored polygons that compose the cutplane. Turning "Faces" off allows better visibility of contour lines and edges if these options are selected.

**Contour Lines**

Below the "Faces" toggle button is a button labeled "Contour Lines." When selected, this button will draw contour lines in the current text color (either black or white depending on the background color) on the cutplane corresponding to the same divisions found in the color bar. For example, if the color has divisions at 5, 10, and 15, then GMV will draw contour lines at these intervals.

**Edges**

If you want GMV to draw the edges of any cells the cutplane intersects, click the mouse on the "Edges" button. There is no need to make sure the option is checked before adding the cutplane. The edges of the intersected cells will instantly appear.

**Height**

The "Height" slider bar allows you to make your cutplane appear three dimensional for greater clarity. The slider bar moves each point on the cutplane away from the plane according to its value as indicated by its color. For example, the red areas, which are the most intense, move away from the plane the greatest distance while the blue areas move the least. To activate the "Height" feature, click on the toggle button on the lower left corner of the "Cutplane" menu. Now drag the slider up and down to adjust the height of the plane. Zero height is in the middle of the slider track.

**Distance**

Next to the Height slider bar in the "Cutplane Options" window is a slider bar labeled "Dist." Next to the slider are boxes labeled "Node Vect.", "Nodes", "Node Nos.", "Cell Vect.", "Cells", and "Cell Nos.". When the slider bar is at the bottom, there are two infinite planes parallel to and on the surface of the cutplane, one on each side. As the slider is dragged up, these planes move away from the cutplane in opposite directions while still remaining parallel to the cutplane. If a node or a cell center lies between the two infinite planes, its corresponding point, vector, or number is displayed. This is the basis behind the cutplane distance function. GMV will display nodes, node numbers, node vectors (if any), cells, cell numbers, cell vectors (if any), or all as the distance is adjusted, depending on which option boxes are highlighted.

**Adding a cutplane to the main viewer (the manual way)**

A cutplane is created by specifying the three noncollinear points needed to define a plane somewhere in space. Choose the "Cutplane" option from the "Calculate" main menu bar and a menu similar to Fig. 4−3 will appear. This menu allows the selection of cutplane creation using a node or cell field, the selection of the desired field in each case, and the creation of multiple cutplanes. To create a cutplane, select an unused number from the lower section of the menu (in this case, "1"). This action will create a menu similar to Fig. 4−4 to allow the definition of the desired cutplane.

Somewhat to the left of this menu and in the center are nine boxes, three rows of three each. Each row defines a point. Enter numbers in the boxes by clicking on the appropriate box and typing in the number. Pressing the tab key will highlight the next box in sequence so that all of the boxes will eventually be filled with numbers. Each row is a point, not each column. When you are satisfied with the points defined, you must choose how to color the cutplane. You can choose either node or cell field values in the first menu (Fig. 4−3). If you choose "Node Field" then each cell will be colored according to its nodes and node field values. If you choose "Cell Field" then each cell will be colored one solid color according to its cell field value. When you have made your choice, click on the "Add" button below the data boxes. If the plane defined by the given points intersects the mesh in any place, GMV will draw a plane there. To remove the plane from the screen, click on the "Delete" button next to the "Add" button. The cutplane will be color coded according to the current node field or cell field, which is displayed on the right side of the window. In Fig. 4−3, the field "pressure" happens to be chosen. This is how color contour plots of data along cutplanes are generated. To change fields, click on the "New Field" button to pop up either the Node Field Selection menu or the Cell Field Selection menu. The "Apply Field Change" button updates the existing cutplane(s) to display the new field information. If multiple cutplanes have been defined, this operation applies the new field to all cutplanes.

Cutplanes may also be temporarily disabled on an individual basis by choosing the particular cutplane from the menu shown in Fig. 4−3, then toggling the "On" button to make the cutplane invisible.

**Adding a cutplane the easy way**

There is a much easier method of adding a cutplane that does not require you to specify three points. This method is called the "2 Points" method and its toggle button is located in the upper left corner of the window. Using this function, a cutplane is defined by clicking the mouse on two points on the screen. The cutplane created by this action goes through the two points and is always normal to the screen.

To define points, first click on the toggle button and move the mouse to the main viewer. The mouse arrow will change into crosshairs. Click with the left mouse button on any two points on the screen. After doing so, GMV will copy these points into the boxes in the cutplane window as well as fill in the missing coordinates needed to make the plane normal to the screen. When the points have been defined, click on "Add" to display the cutplane.

## Distance:

The "Distance" function in the "Calculate" menu is used for calculating the linear distance between two points along the screen. To use the distance function, first choose it from the menu. As soon as the function is chosen, the mouse pointer will change into crosshairs resembling a plus sign when it is in the main viewer. Click the crosshairs on one of the two points between which the distance is to be calculated. A small grey dot will appear. Move the mouse to the other point and click the crosshairs there. After both points have been defined, GMV will display the distance in the upper portion of the main GMV window to the right of the background color controls. Each time a distance is calculated, the old number next to the color controls will be overwritten with the new one.

## Field Calc.

The "Field Calc." option of the "Calculate" menu is used to calculate data from an existing node field or cell field using supplied unary or binary operations. The calculated data is placed in one of the five extra fields provided.

**Selecting a field to build**

Choose the "Field Calc." option from the "Calculate" main menu bar then select either "Node Field" or "Cell Field" (if available) and a menu similar to Fig. 4.6 will appear. The "FldCalc" names indicate that data for the field has not been calculated. Select one of the five fields and a menu similar to Fig. 4−7 appears.

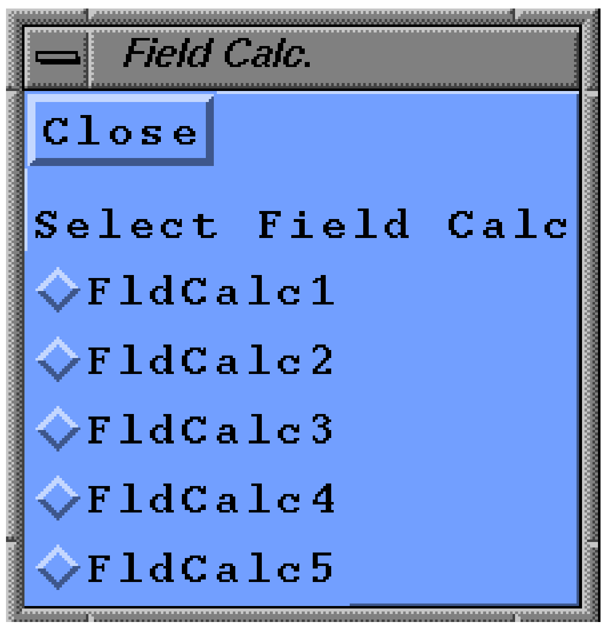



***Figure 4−6. Field Calc. Selection menu***

**Build (calculate) the new field**

To build the new field, first select an operator from the

Operators list. Note that the operators refer to the variables X, Y, and C. The X and Y variables refer to selected fields, while the C variable represents a user defined constant. Next, fill in the constant field "C=", if needed, and check that the current fields displayed the "X Field" and "Y Field" are the fields to operate on. If

not, click on the appropriate "New Field" button to select a field from the Field Selection menu. Finally, enter a name in the "New Field Name" text field and click on the "Add" button. If an invalid operation, such as the square root of a negative number, is recognized for a field data element, a warning message will appear on the menu and that data element will not be changed. The new field is now available for use as is any other field and is identified by the new field name. Use the "Delete" button to delete the selected calculated field. When a calculated field is deleted, the field name changes to the "FldCalc" prefix.

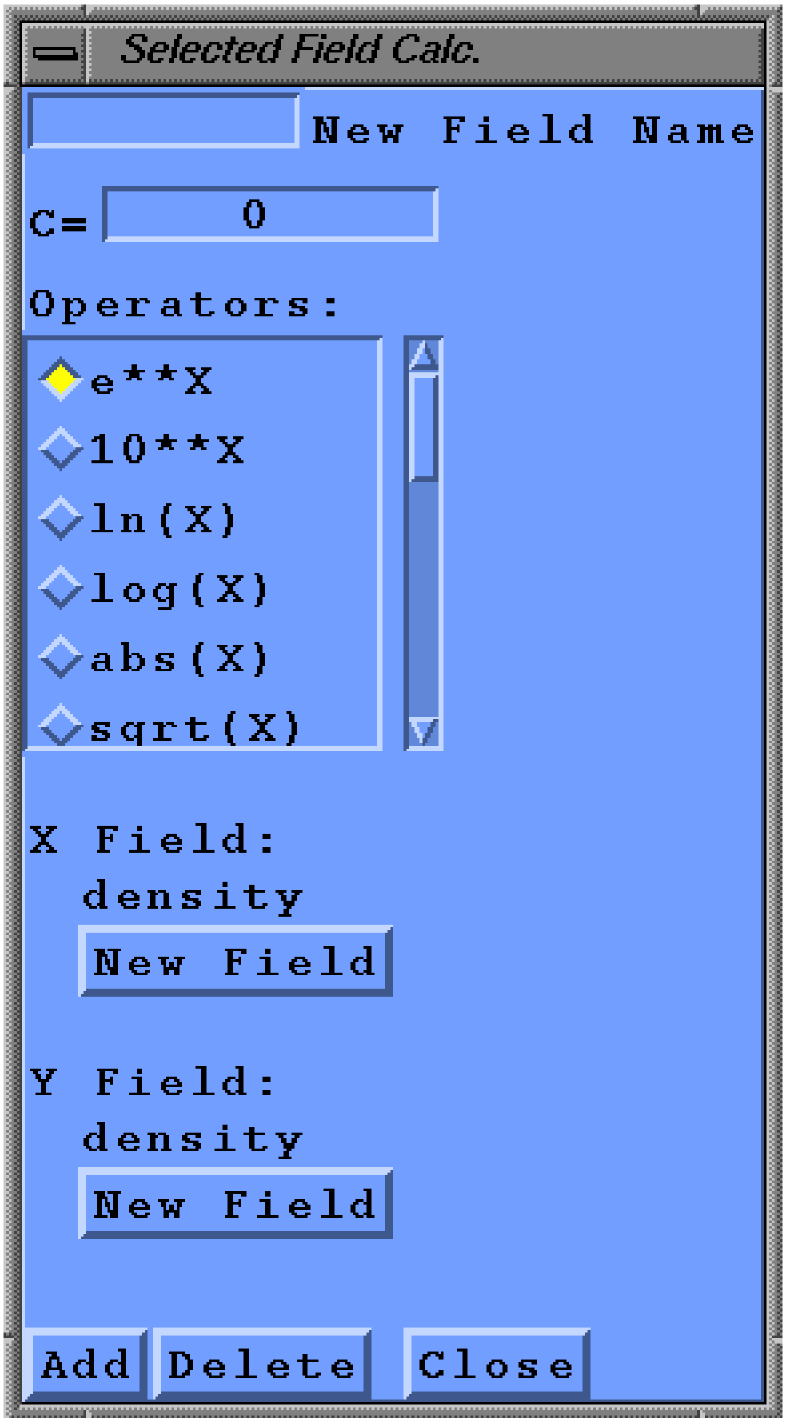



***Figure 4−7. Field Calc. Build menu***

## Grid Analysis:

"Grid Analysis" allows you to selectively view only a portion of the mesh data contained in a GMV input file. This is done by specifying individual nodes or cells to be viewed on the screen. This function is especially useful when there are thousands of nodes and cells in a given set of mesh data. The ability to isolate individual areas of data allows for more detailed analysis of the problem at hand. To open the grid analysis window, use the mouse and choose "Grid Analysis" from the "Calculate" menu. A menu similar to Fig. 4−8 will appear.

### Selecting cells by nodes or cell numbers

Cells can be selected by cell numbers or by specifying nodes that are common to one or more cells. In the window are two columns of eight boxes each used for data entry. The first column is labeled "Select cells by nodes." Here you can give GMV the numbers of a few of the nodes. Based on this information, GMV will decide which cells are relevant and display them on the screen. GMV will display the cell that contains all of the given nodes as vertices. Nodes that are not vertices of any cell are ignored by GMV. If two or more cells contain the same number of the given nodes, GMV will display all of the relevant cells. The second column of data boxes is labeled "Select cells by number." Here you may list the numbers of the cells to view. Fill as many boxes as necessary with the relevant cell numbers (there are a maximum of eight boxes). To view a specific range of cells, such as cells numbered from 1 to 50, choose cells by placing the numbers of the first and last cells of the series in the boxes at the bottom of the submenu. These boxes are labeled "Begin" and "End." GMV will show all of the cells in that particular series.

After all of the node and cell selection data has been entered into the appropriate boxes, direct GMV to display the requested


*Figure 4−8. Grid Analysis menu*

cells. There are two buttons at the top of the menu that accomplish this. They are labeled "Start New Display" and "Add to Display." If the "Start New Display" button is used, GMV will erase any objects the grid analysis function created beforehand, before it draws in the selected cells. If the "Add to Display" button is used, GMV will add the newly chosen cells to the display. The cell drawing commands in the "Grid Analysis" menu are the same as those for the "Cells" display menu with the addition of "Node Numbers". These drawing commands only apply to nodes and cells selected by grid analysis. All other drawing options are still available.

Note that the "Grid Analysis" menu is created with the "Edges" button selected as the default. When nodes or cells are specified, the default behavior is to draw the cell edges in the display. The node numbers of the cells of interest are only displayed when the "Edges" button is selected. As such, if node numbering is desired but does not appear as expected, ensure that the "Edges" button is activated.

To exit "grid analysis", click on the button labeled "Exit" in the upper left corner of the menu. This action deletes the grid analysis cell drawing and closes the menu.

**Color By:**

Cell faces and edges are colored the same way as in the "Cells" menu under the "Display " option.

## Isosurfaces:

An isosurface is the surface in space where a selected field is always a particular value. For example, the isosurface where the voltage around a charged particle is constant would be a spherical shell. Isosurfaces for any field in a GMV input file can be generated using this tool, (see Figs. 4−9 and 4−11).

**Adding a material isosurface**

GMV can generate

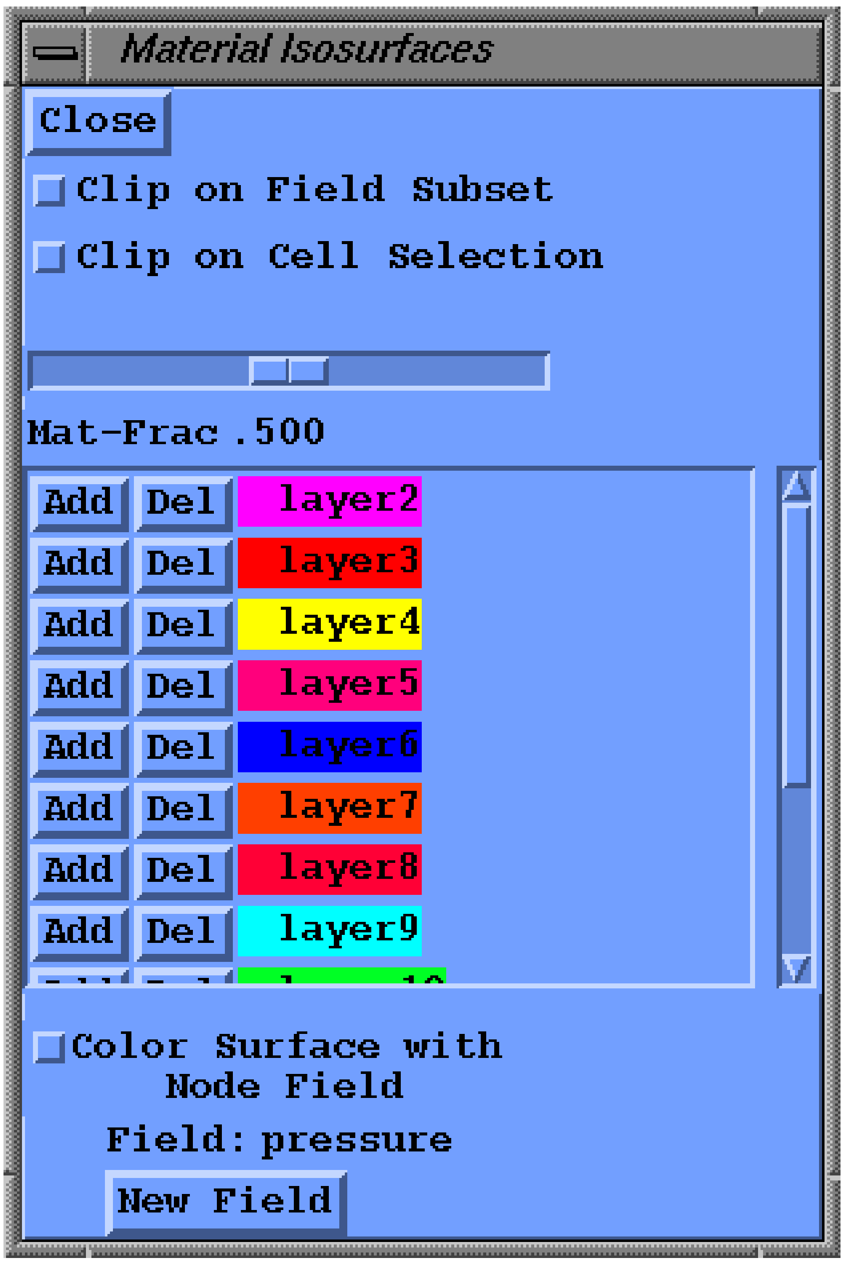



*Figure 4−9. Material Isosurface menu*

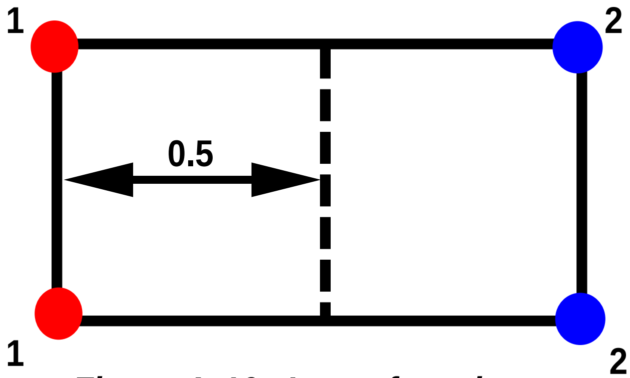



*Figure 4−10. Isosurface demo.*

an isosurface from node material data. Because of the noncontinuous nature of material data, there is a special method for drawing these surfaces. Even if the material data is cell centered, each node is assigned a material number. Click on "Calculate" from the main menu bar. Click on "Isosurfaces." A submenu with two options will appear to the right. Choose "Materials" to bring up the "Material Isosurfaces" menu, (see Fig. 4−9). To add a material isosurface, first adjust the material fraction ("Mat. Frac.") slider bar above the list of materials. The material fraction tells GMV how far between differing materials to draw the surface. For example, take a rectangle with four nodes. Two of the nodes are material number 1, and the others are material number 2. With a material fraction of 0.5, GMV would draw the isosurface halfway in between the two different materials, (the dashed line in Fig. 4−10). After the material fraction has been adjusted, click on "Add" next to the box corresponding to the desired material to add the isosurface. Click on "Del" to remove the surface. Note: truer material surfaces can probably be generated by simulation code and entered as surface polygons.

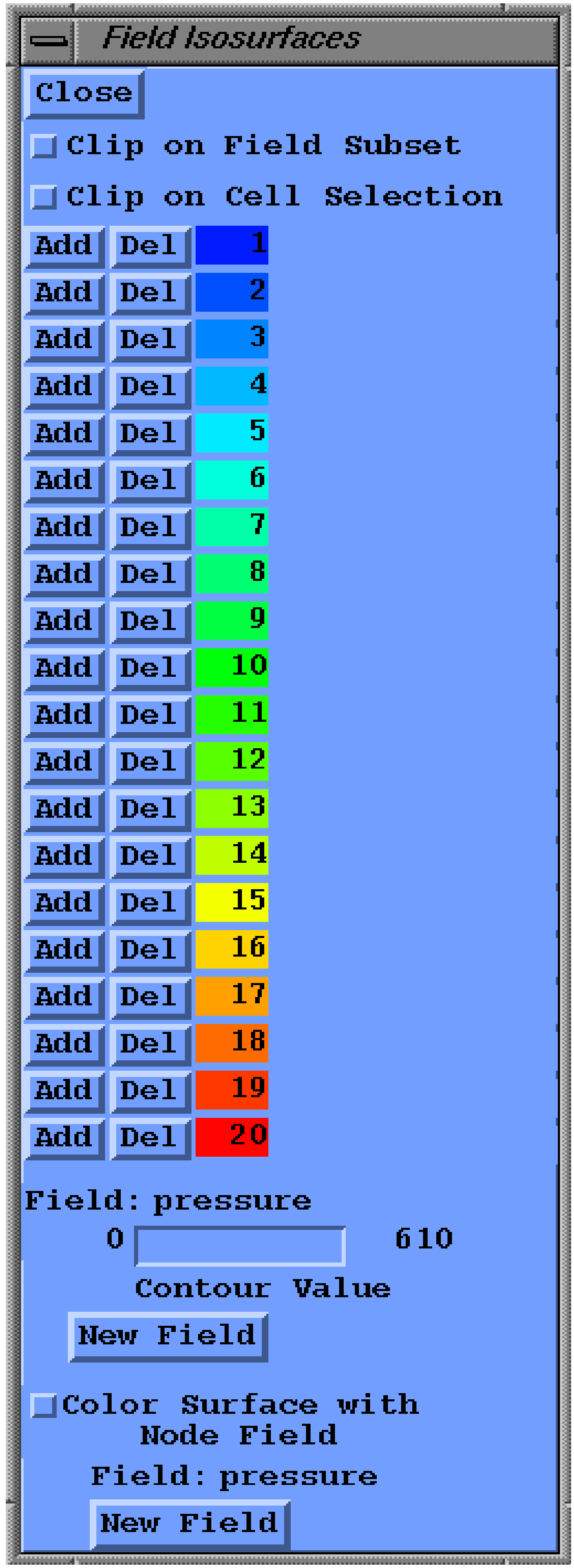



*Figure 4−11. Field Isosurface menu*

### Adding a field isosurface

Open the "Field Isosurface" menu from the main menu bar under "Isosurfaces" in the "Calculate" menu. A menu similar to Fig. 4−11 will appear. To add a field isosurface, first check that the desired node fields is the current field. The current node field is displayed after "Field:" near the bottom of the "Field Isosurfaces" menu. To change the current field, click on the "New Field" button to pop up the Node Field Selection menu Below the current field is a data entry box into which a contour value must be entered. The "Contour Value" is the number in the selected field that will generate the surface with that value. To enter a number, click on the box and enter a value between the minimum and maximum values for the selected field. The range of min. and max. values appears on the right− and left−hand side of this particular box. Finally, you must choose a color for the isosurface. Down the middle of the "Field Isosurface" menu is a column of colored boxes numbered one through twenty. Choose a color for the isosurface. Click on "Add" next to the desired color. GMV will beep and display the isosurface. If the contour value is not within the range specified on the sides of the contour value data box or if the isosurface for a particular value simply does not exist, GMV will show nothing. GMV lists the field and contour value of each isosurface next to the surface's color box in the isosurface window. To delete an isosurface, click on "Del" next to the appropriate isosurface color box. Isosurfaces can be turned on and off by clicking on the colored box.

### Clip on field subset and cell selection

These two toggle buttons tell GMV to draw isosurfaces based only on data contained within some subset. For the "Clip on Field Subset" option, the subset is defined by the "Subset" function. The "Subset" function is found in the Controls−2 menu. This function is explained later on in this text. For the "Clip on Cell Selection" option, the subset is defined by the Boolean expression created using the "Select" option in the "Cells" menu. Isosurfaces will then only be drawn from node data found in the selected cells.

GMV only checks the status of these two buttons when an isosurface is created. Therefore, any changes made with these two options will not take effect until a new isosurface is calculated.

### Coloring isosurfaces with field values

The isosurfaces can be colored with the blue−to−red intensity color of node field values. All the isosurfaces for a specific type (material isosurfaces or field isosurfaces) are colored accoridng to the selected field values. To color the isosurfaces with field values, first select a field with the "New Field" button, then click on the "Color Surface with Node Field" button.

## Isovolumes:

The "Isovolume" option of the "Calculate" menu allows the creation of isovolumes (the display of a volume with surfaces of which are interpolated between the selected minimum and maximum field values). The selection of this option creates the menu shown in Fig. 4−12.

This menu lists the current active field, and the minimum and maximum values of this field. To add an isovolume, enter the value for the minimum and maximum surfaces in the "Isovolume Range" areas, then select "Add." The isovolume may be toggled on and off using the "On" button. The "Clip on Field and Cell Selection" functions behave similarly to the same buttons described for the "Isosurfaces" menu. The isovolume surface can also be colored using node field values as described for isosurfaces.

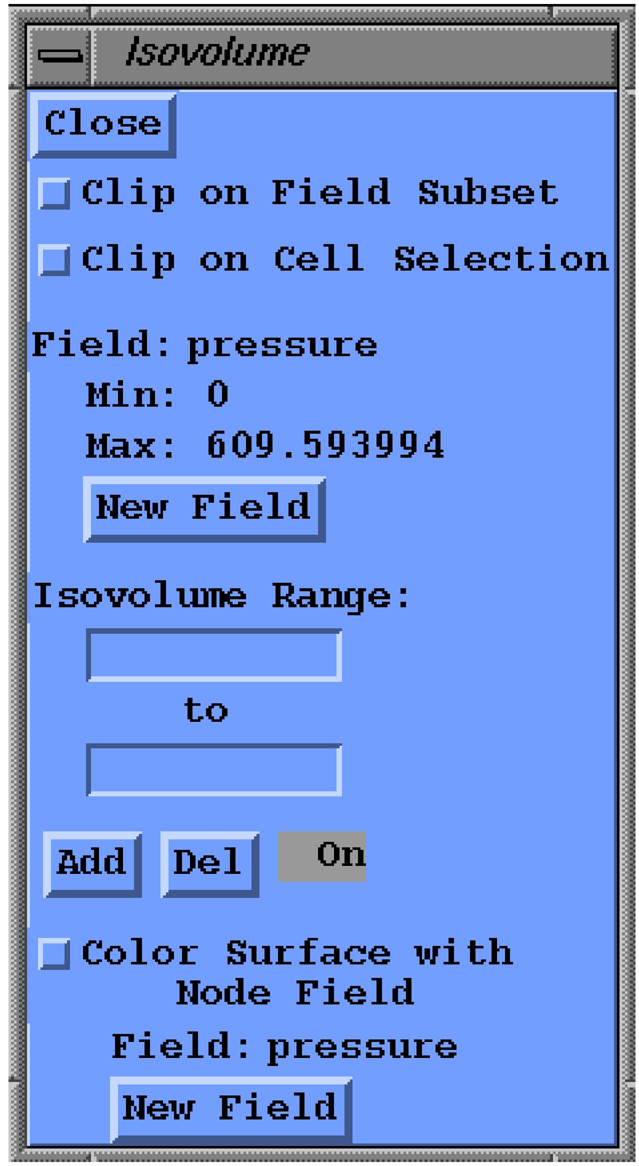



***Figure 4−12. Isovolume menu***

## Query Data:

The "Query Data" option of the "Calculate" menu makes it easy to find information about any cell or node in the grid. Query Data also has facilities to interactively probe the image for a node or cell number. There is a function to obtain node numbers by node field value.

### Getting node and cell values

Choose the "Query Data" option from the "Calculate" main menu bar and a menu similar to Fig. 4−13 will appear. The menu has two main parts. The left side displays information about a selected node and the right side has information about a selected cell. To list all the data associated with a specific node, enter the number of the node in question in the box labeled "Node:" and click on the "Get Values" button. All of the available information about the specified node will appear below. The first three numbers are the node's X, Y, and Z coordinates. The three numbers after that are the magnitudes of the i, j, and k velocity vectors if any exist, which are labeled U, V, and W respectively. After the velocity, GMV will list all field values for the node (i.e. temperature, pressure, or speed data). Finally, GMV lists the node material and flag values.

The "Cell Info" side of the window can be used to retrieve information about individual cells. To list the data associated with a specific cell, enter the number of the desired cell into the box labeled "Cell:" and click on the button labeled "Cell Values." All of the available data for that cell will be displayed below the box with the cell number in it. First is the cell center's X, Y, and Z coordinates followed by any cell centered field data, then the cell material and flag values followed by the numbers of all the nodes that make up the vertices of the cell. Note that any cell−centered data is also displayed as a node value. GMV averages all cell−centered data to the nodes. Any data defined to be cell−centered in the input file will be averaged to the nodes and be displayed on the node value side of the grid value window.

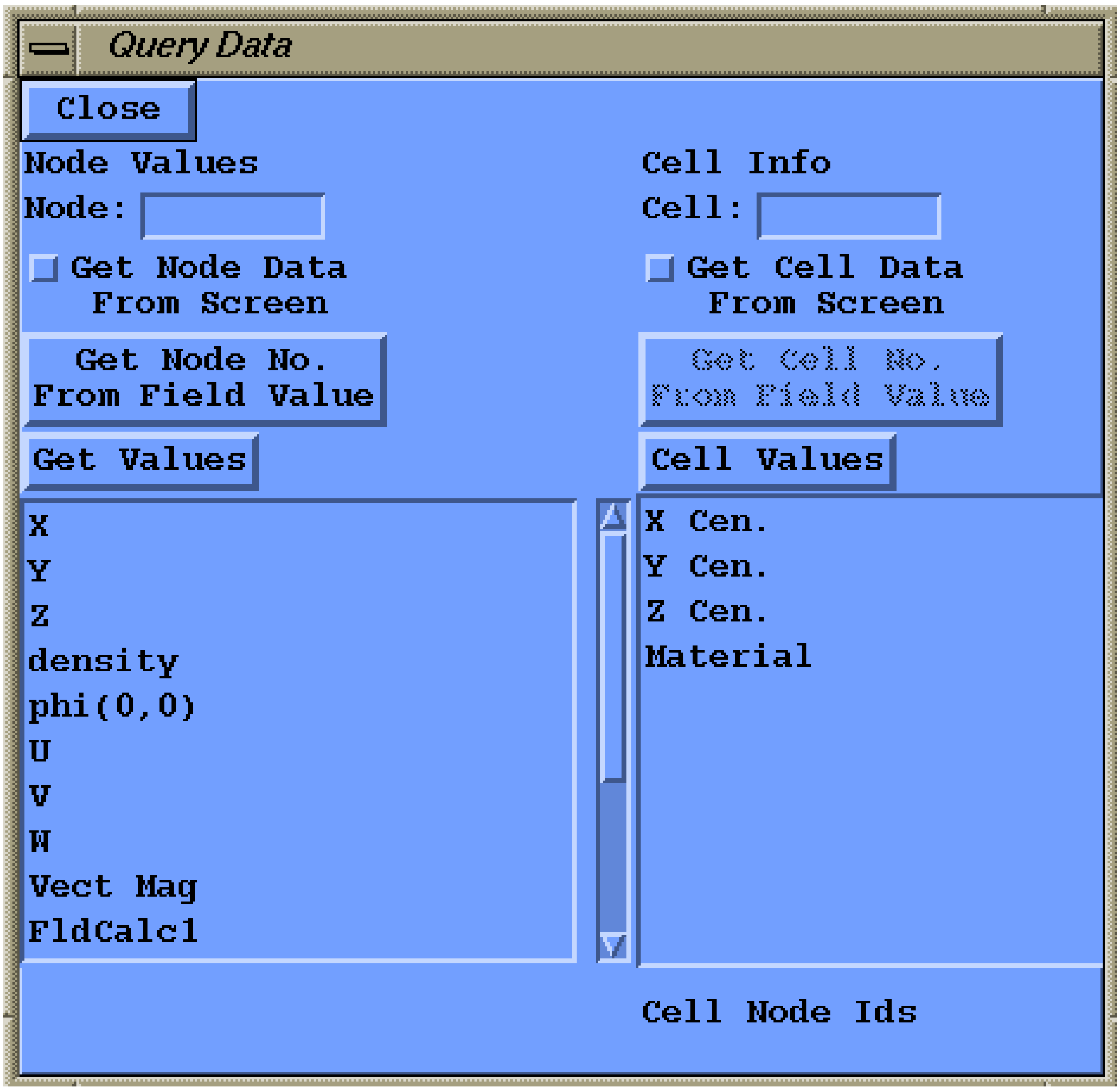



***Figure 4−13. Query Data menu***

**Probing node and cell numbers from the image**

The "Get Node Data From Screen" button allows you to use the mouse to get node data from a node displayed on the screen. This feature is available whenever nodes are displayed by the "Nodes" display option. Click on the "Get Node Data From Screen" button and move the cursor to the image, then move the crosshair cursor to a node and press the left mouse button. The number of the node closest to the crosshair will be displayed in the "Nodes:" box. Also, the data for the node will be updated. You may continue to probe nodes until you click off the "Get Node Data From Screen" button . Similarly, the "Get Cell Data From Screen" button returns the cell number and data of the cell whose cell center is closest to the crosshair when cells are displayed.

**Getting node and cell numbers by field value**

You can retrieve a node number by node field value by clicking the "Get Node No. From Field Value" button. When selected, a menu similar to Fig. 4−14 appears. Select a field to query and the field data minimum and maximum are displayed along with the first node number at these extremes. If there more than one node with a minimum or maximum value, the number of matches is displayed. To get the node number closest to a specific value for the current field, enter the value in the text box labeled "Match Value" and click on the "Get Node" button. The node number closest to the match value is displayed along with the exact value of the

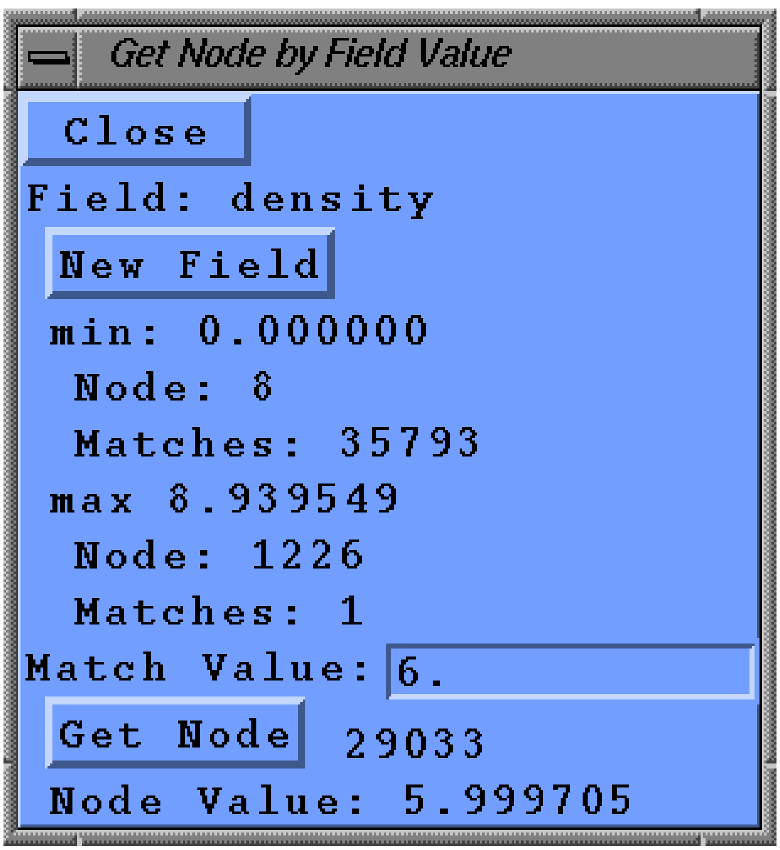



***Figure 4−14. Get Node by Field Value menu***

# The Controls−1 Menu

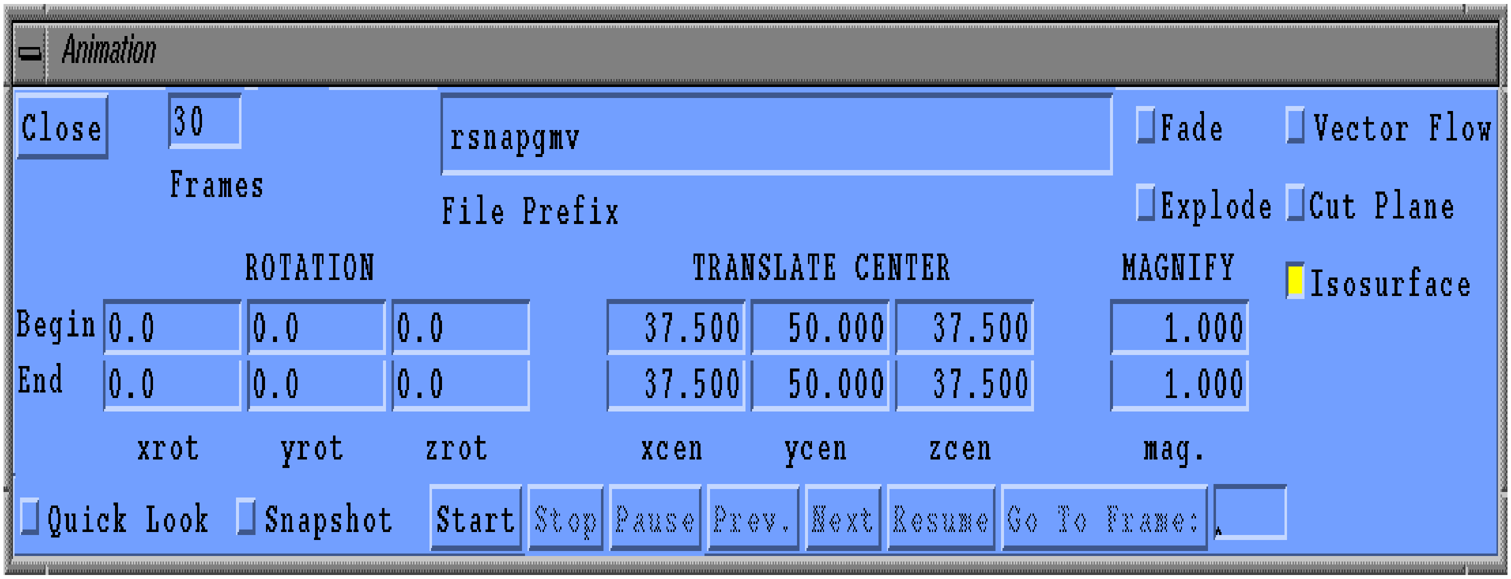



***Figure 5−1.  Animation menu***

## **<u>Animation</u>** (orthographic and perspective modes)**:**

GMV can be used to create animation sequences.  To open the animation menu, choose the "Animation" option from the "Controls−1" main menu bar.  There are two different animation menus, depending on which "View" mode you are currently in.  If you are in orthographic or perspective modes, a menu similar to Fig. 5−1 should appear.  To begin an animation, click on the "Start" button.  Click on "Stop"  to stop the current sequence.  A given animation sequence may be paused at any time during its run by clicking on the "Pause" button.  When paused, the animation sequence may be stepped through one frame at a time by clicking on the "Prev." and "Next" buttons located at the bottom of the menu.  You may jump to a specific frame by entering its number in the box with the "Go To Frame" button next to it and clicking on that button.  To resume the animation where you left off, click on the "Resume" button.

**Number of animation frames**

In the upper left corner of the "Animation " menu is a box labeled "Frames."   Enter the number of frames the sequence should contain.  Thirty is the default (approximately one second of animation on video).  The more frames there are, the smoother the animation sequence.

**Rotation**

The object in the main viewer can be rotated during the animation sequence by specifying an angular distance in degrees for the object to move. In the left center of the animation window are six boxes with the heading "Rotation." The top row of three boxes is labeled "Begin" and the bottom row of boxes is labeled "End." By specifying beginning and ending angles in the appropriate boxes, the object can be rotated about any axis relative to the current view angles. For example, if you want to begin at 90 degrees of rotation and end at 180 degrees, then enter 90 in the top and 180 in the bottom of the X−rotation column. This would rotate the object in the main viewer about an axis going through the center of the object parallel to the X−axis, exactly 90 degrees. The same is true for the Y and Z rotation columns. X, Y, and Z rotations can be done simultaneously by entering the appropriate numbers in the boxes, but this process is not recommended. If no rotation is desired, enter the same number (usually zero) in both the begin and the end boxes to achieve this effect.

**Center translation**

The object in the main viewer can be moved linearly in any direction during an animation. This motion is accomplished by entering data into the set of six boxes with the header "Translate Center." The first row of three boxes is used to tell GMV where to start the center translation in the form of X, Y, and Z coordinates. The second row tells GMV where to end the movement. GMV takes default values from the "Center" tool also found in the "Controls−1" menu.

**Magnification**

The magnification of the object in the main viewer can be increased or reduced during an animation sequence. On the far right of the animation window are two boxes with the header "Magnify." In the top one enter the starting magnification and in the bottom box enter the ending magnification. During the sequence, the object will either grow larger or smaller depending on whether the beginning magnification is greater than or less than the ending. If no change in magnification is desired, enter the same number into both boxes (usually 1.00). The object will then remain the specified size throughout the entire animation sequence.

**Vector flow**

The vector flow option displays the paths that particles with vector data would take. GMV shows the vector flow by dividing the length of the each node’s current vector into as many equal pieces as there are frames and then displaying each piece in sequence during the animation. To include vector flow in an animation, turn vectors on and then click the toggle button in the upper right corner of the animation submenu before starting the animation.

**Cutplane**

The animation window includes provisions for sweeping a cutplane across the object in the main viewer. The cutplane may sweep perpendicular to the X, Y, or Z axes. Only one sweeping cutplane may be in an animation at a time. To insert this effect into the animation, click on the box with the label "Cut Plane" in the upper right corner of the animation submenu. A submenu similar to Fig. 5−2 should appear. The first four options allow choosing along which axis the cutplane should sweep. GMV can only sweep across one axis at a time. Clicking on the "None" option will turn off the cutplane option. Below the axis options are two slider bars. These sliders control where the cutplane sweep begins and ends along the chosen axis. The default is to sweep the entire length of the object. During an animation sequence that includes a sweeping cutplane, the coordinate on the selected axis where the current cutplane lies will be displayed at the bottom of "Cut Plane" menu. Below the node and cell field label is the currently selected field for interpolation onto the cutplanes. Click on the "New Field" button to select a new field from the Field Selection menu. When all the selections have been made, click on "Close" to remove the submenu from view.

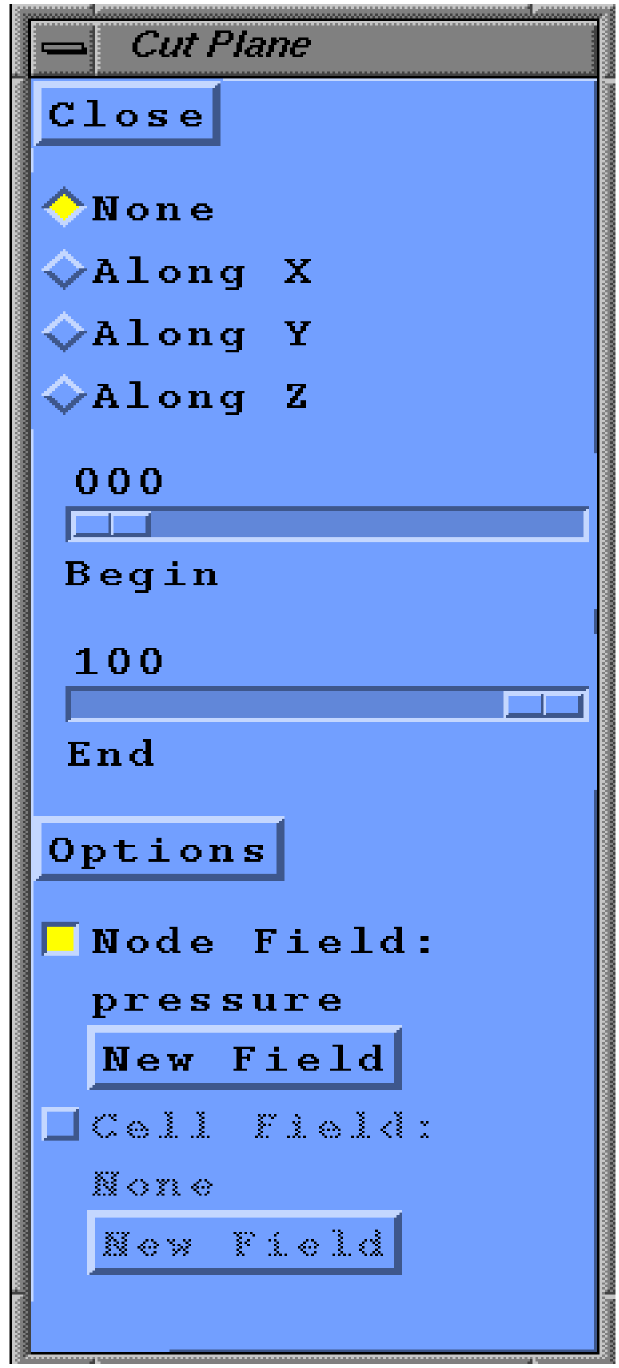



***Figure 5−2. Cutplane Animation submenu***

**Fade**

The fade option allows for fading out polygons and isosurfaces over the course of an animation sequence. To use the fade option during an animation, click on the toggle button labeled "Fade" in the animation menu. A submenu similar to Fig. 5−3 will appear. First, set the beginning and ending transparency factors using the yellow slider bars at the top of the submenu. A value of one corresponds to no transparency and a value of zero is 100% transparent. Next, choose the material or isosurface to be faded during the animation by clicking on the appropriate box. When a selection is made, the beginning and ending transparency factors will

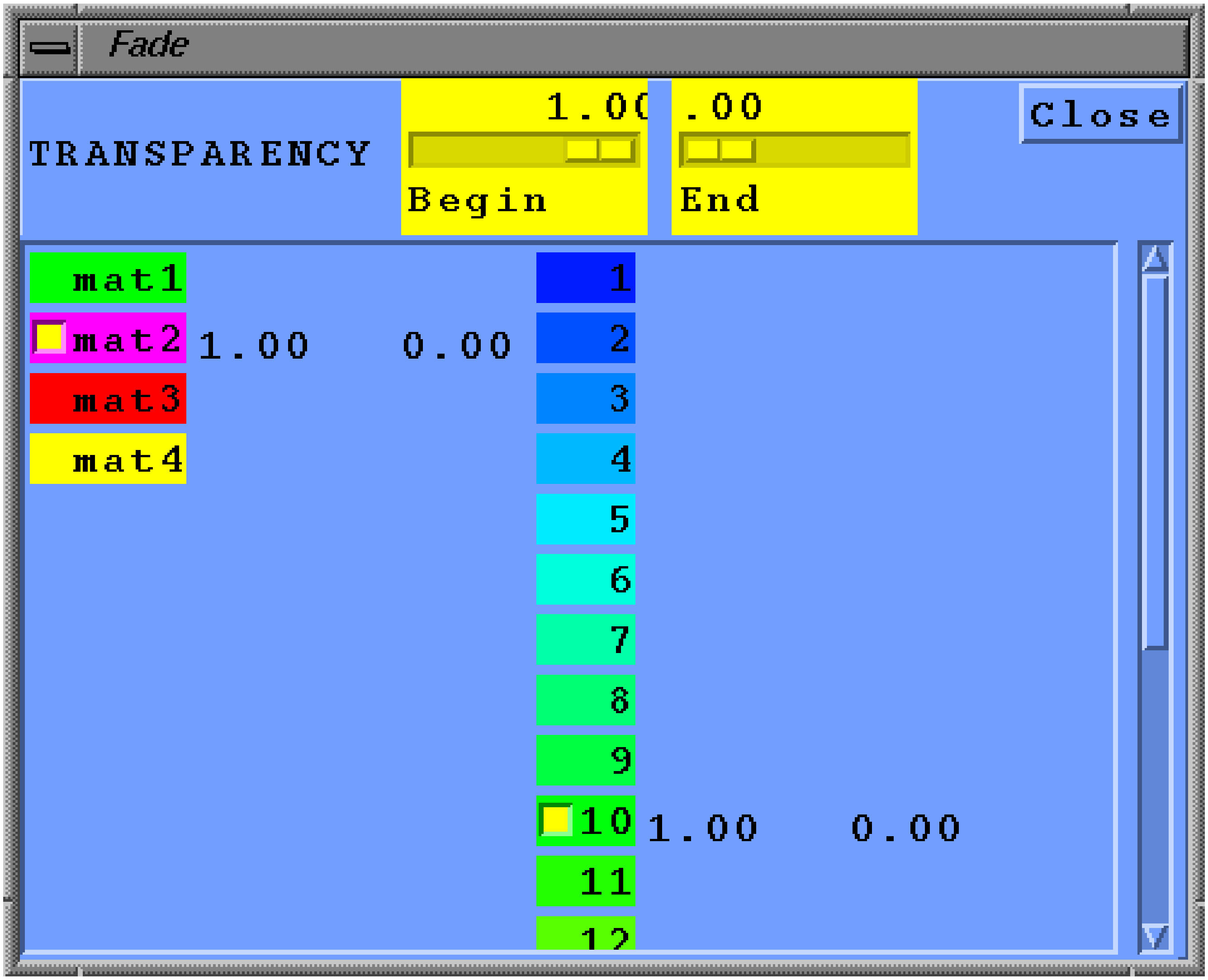



***Figure 5−3. Fade Animation submenu***

appear to the right of the box. When finished choosing things to fade, click on "Close" to remove the submenu from view. The toggle button in the animation submenu will remain on, indicating that fading will be included in the next animation. To turn off "Fade," toggle all of the material and isosurfaces in the fade submenu off. Now, when the window is closed, the fade option will also be toggled off.

**Exploding cells or polygons during animation**

You can explode polygons and cells during an animation sequence. Toggle the button labeled "Explode" in the animation menu. A submenu similar to Fig. 5−4 will appear. With the two slider bars at the top, adjust the beginning and ending explode percentages. Next choose polygons and/or cells to explode. The beginning and ending explode percentages will appear to the right of whatever was chosen to explode during the animation. For cell explode, choose material or the flag to separate the cells. Click on "Apply" to activate your choices. After choosing the various options, click on "Close" to remove the submenu from view. The "Explode" toggle button in the animation menu will remain on until both "Polygons" and "Cells" are toggled off in the explode window.

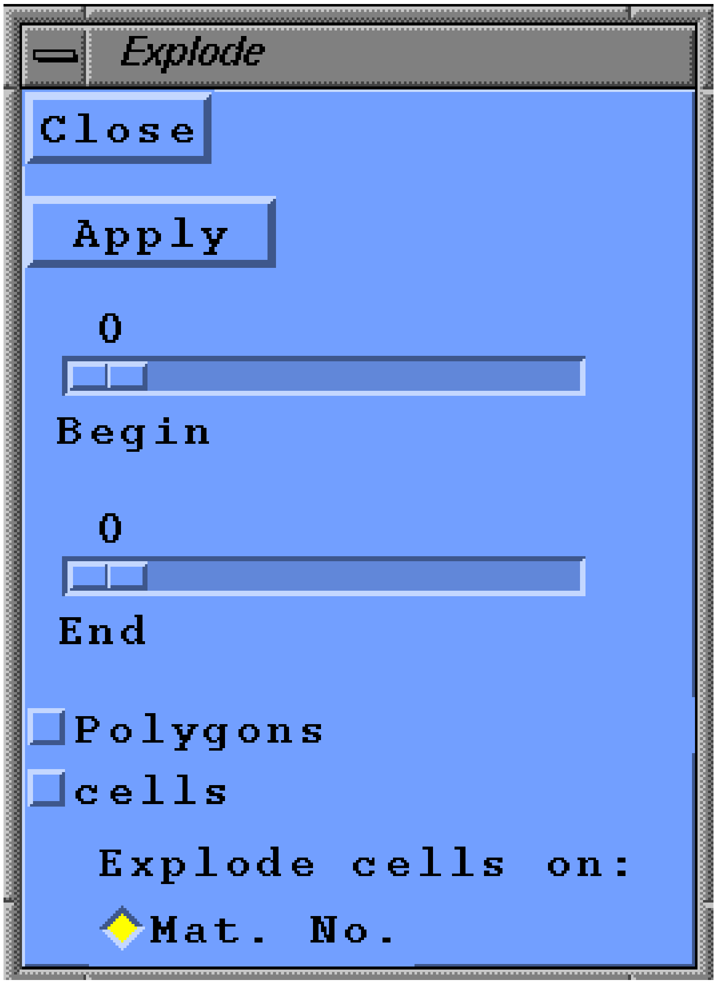



***Figure 5−4. Explode Animation submenu***

**Snapshot**

Each frame of an animation sequence can automatically be saved using the snapshot function in the animation menu. Highlight the "Snapshot" option to the left of the "Start" button. When this box is highlighted, GMV will save each frame of the animation sequence in its own file. All of the files from the same animation will have the same prefix. To enter another prefix into the box labeled "File Prefix" at the top of the animation menu, click and type in a prefix. At the end of the file prefix, GMV will append the frame number of that particular snapshot. For example, the seventh frame might have the name "snapshot007" with the 007 indicating that the file contains the seventh frame of the sequence. The animation snapshots are created in movie size (640x512) so they can be combined with time sequence images to create movies. Note: the "Pause" button is not active while saving snapshots.

**Quick look**

The "Quick Look" option tells GMV whether or not to use the current interactivity setting during the animation. The interactivity can be adjusted with the slider bar on the right of the main GMV window labeled "Int." Without "Quick Look" selected, the animation will be carried out with all objects drawn. This option can speed things up, especially when many objects are drawn.

### Isosurface animation

Isosurfaces may be added to an animation sequence by toggling the "Isosurface" button in the "Animation" menu.  A submenu similar to Fig. 5−5 will appear.  The first two radio buttons turn the isosurface animation on or off.  Click "On."  Underneath the on and off buttons is the current field from which isosurfaces can be calculated.  Click on the "New Field" button to select another node field from the Node Field Selection menu.  After a field is chosen, GMV places the default data range for that particular field in the "Begin" and "End" boxes near the bottom of the window.  The default data range is defined as the minimum and maximum values in the current field.  To change the data range, click on either the "Begin" or "End" data box and enter the desired value.  GMV will divide the data range into as many equal increments as there are frames.  During the animation sequence, a new isosurface will be drawn for each contour value in the data range, one for each frame.  Also during the animation sequence, the current isosurface contour value will be displayed at the bottom of the "Isosurface" menu.  The color of the isosurface animation can be changed using the "Color Edit" function in the "Controls−1" menu.  Click on "Close" to remove the submenu from view.  The next time an animation sequence is started, isosurface animation will be included.  To turn off isosurface animation, reopen the isosurface submenu and click on "Off."  When the window is closed, the isosurface button in in the the animation menu will not will not be highlighted.

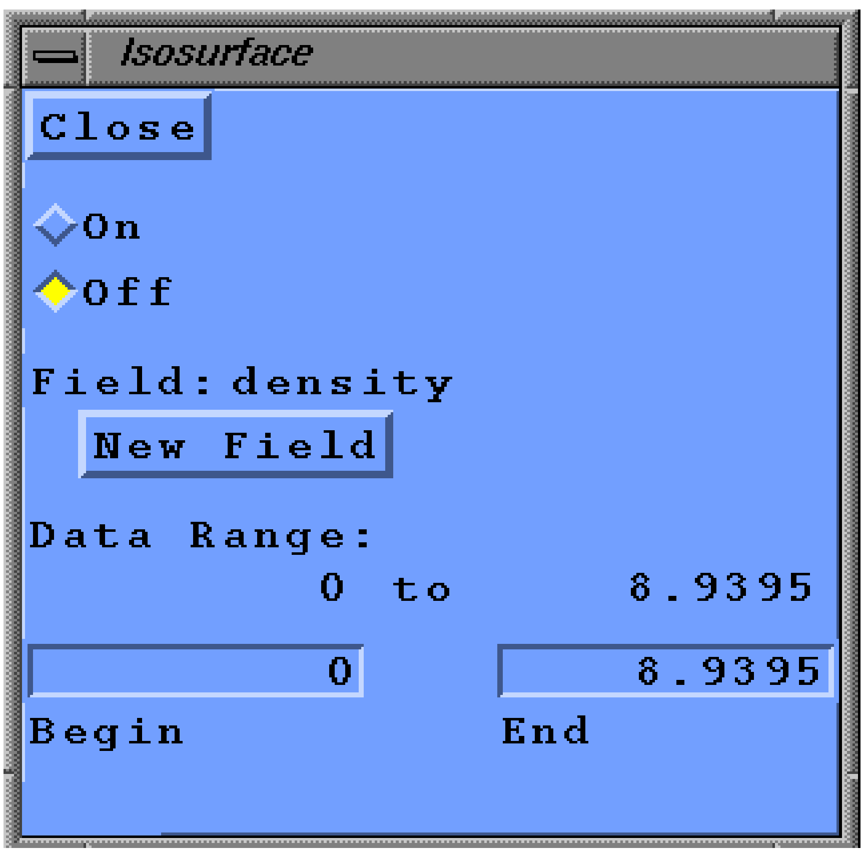



***Figure 5−5.  Isosurface Animation submenu***

## Animation (flight mode):

In flight mode, an animation menu will appear. (See Fig. 5−6) Flight animation moves the simulated helicopter from point to point in space, going up, over, around, and through the object in the main viewer. To start an animation click on "Start." To stop the animation, click on "Stop."

**Setting control points**

Control points are points in space that define a path for the flight mode simulated helicopter to follow. In order to set control points, one should be familiar with the flight mode mouse controls. For a review of these controls, please refer to the **View Menu** section.

To set a control point, first select the number of frames wanted for each control point. The default is 30, but it can be changed by clicking on the box labeled "Control Point Frames" and entering a new number. Next, move the simulated helicopter to the location of the first control point using the mouse. Click on "Set New Control Point" when the location is reached. The number of frames for that control point will appear next to the control point's number in the list at the bottom of the window.

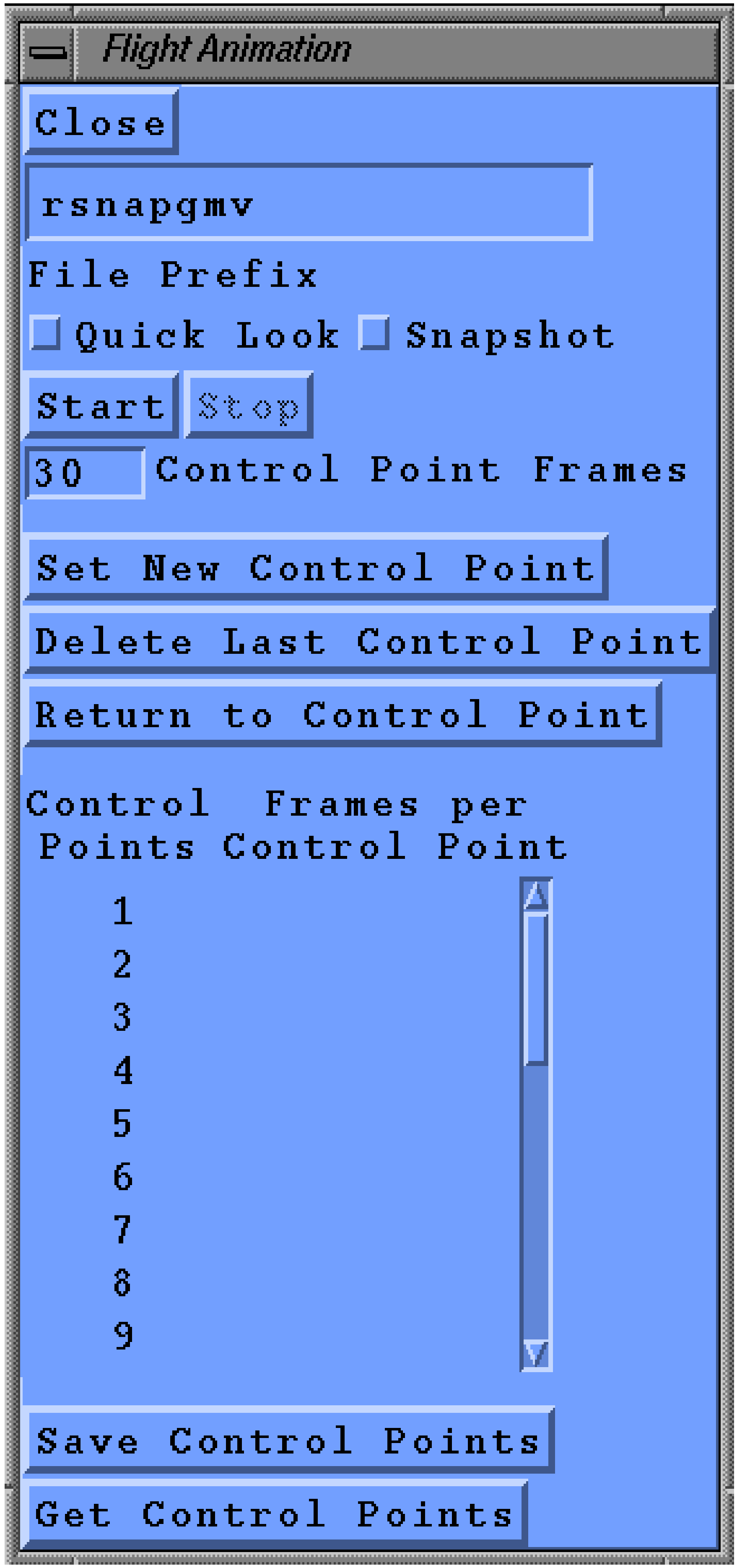



***Figure 5−6. Flight Animation menu***

When all control points have been set , click on "Start" to begin the animation.

To delete a control point, click on the button labeled "Delete Last Control Point." This will clear the most recently entered control point. Control points must be deleted sequentially from last to first and cannot selectively be removed by number.

Should a mistake entering a control point be made, there is a way to return to the most recently set control point so that, by retracing steps, one can try again. To go back to the last control point, click on "Return to Control Point."

**Saving and Retrieving control points**

At the bottom of the "Flight Animation" menu are two buttons. These two buttons, labeled "Save Control Points" and "Get Control Points" allow you to save and retrieve control points. Depending on which you choose you will enter various menus.

**Quick look and Snapshot**

If the object in the main viewer is large and complicated, you can choose to incorporate the current interactivity setting into the animation. Click on the "Quick Look" toggle button.

GMV can also take snapshots of every frame during the animation sequence. To do this, click on the "Snapshot" toggle button and enter a file prefix into the box labeled "File Prefix." Each snapshot file will then begin with the same prefix followed by a three digit number indicating the frame number of that particular snapshot. All snapshots are in Silicon Graphics RGB format.

## Axes:

"Axes" is the second option in the "Controls−1" menu. Its function is to turn on or off the X, Y, and Z axes in the main viewer. To activate this function, choose it from the menu. The axes will appear or disappear depending on whether or not the axes are currently showing. Turning off the axes in the main viewer will have no effect on the axes orientation view box located in the upper right corner.

## Bounding Box:

The bounding box is a six−faced volume that encloses data in the main viewer. It can be turned on or off. By default, the box is set to be the smallest volume that will enclose all of the data in the main viewer. However, the size of the bounding box can be changed using the "Bounding Box" option. A menu similar to Fig. 5−7 will appear. To turn on the bounding box, click on the box labeled "On" in the lower−left corner. Next to the "On" button for the bounding box is the "Show Coordinants" button. This toggle button allows you to turn the coordinants on or off depending on your preference. The length, width, and height of the bounding box are colored red, green, and blue, respectively.  Next to the point where the three colored lines meet are the coordinates for this point, which is the origin of the bounding box.  There is only one number on the opposite ends of the colored lines. Only the coordinate that is different from the origin coordinates is listed here.  For example, on the far end of the red line, only the X coordinate is listed because it is

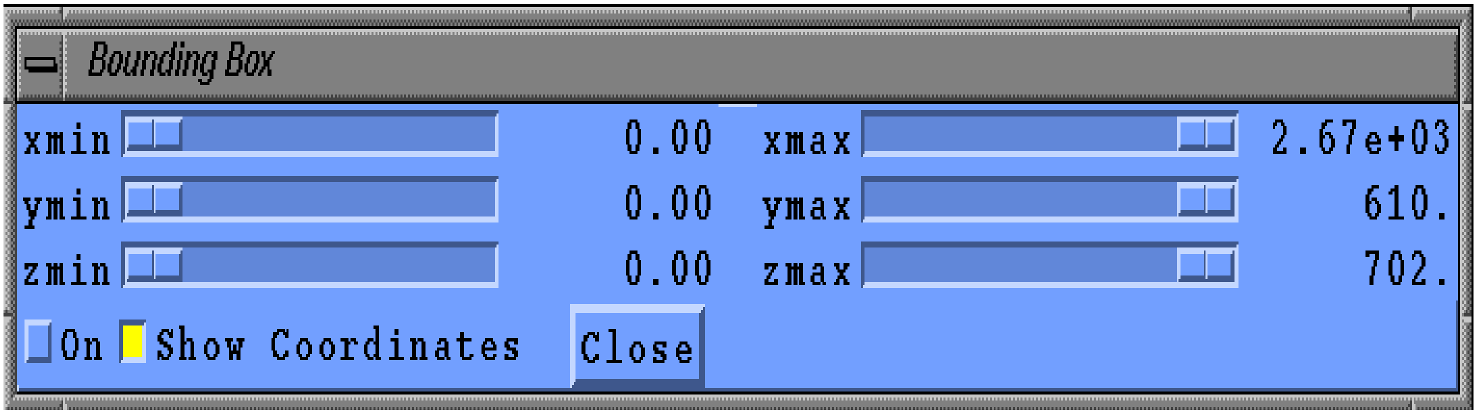



***Figure 5−7.  Bounding Box menu***

the only one that is different from the origin coordinates.  With the numbers provided, it is easy to determine the size of the object in the main viewer.  The size of the box is changed by adjusting the six sliders in the menu.  There is a slider bar for the minimum and maximum values for each axis.  As you slide the bar back and forth, you will see the box change size.  The bounding box cannot grow any larger than the 3−D plotting box.    When finished adjusting the bounding box, click on "Close" to shut the menu.

## <u>Center:</u>

The centering tool is used to tell GMV where the center of the current object lies.  The center is important because it determines how GMV rotates the object in the main viewer.  The values found in this tool also affect any center translation done during an animation.  To open the "Center" window, choose the

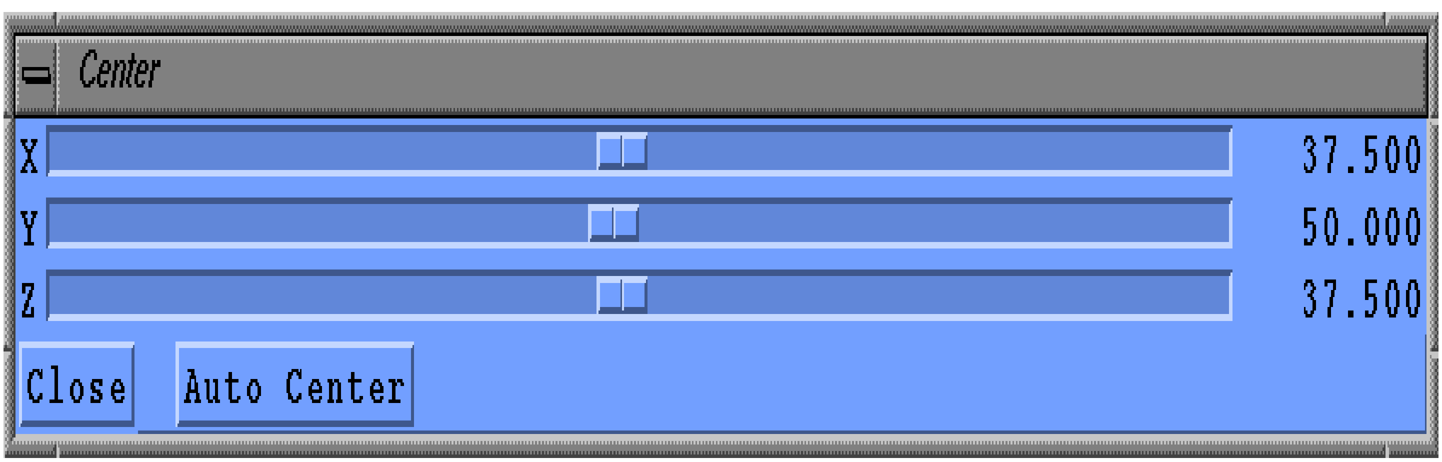



***Figure 5−8.  Center menu***

option from the menu.  A submenu similar to Fig. 5−8 appears. There are three slider bars labeled X, Y, and Z, respectively. Sliding these bars in either direction changes the coordinates of the center of rotation.  On the right of each slider are numbers corresponding to the current coordinate for that axis.  These numbers will change as the slider is moved.  By default, GMV calculates the most convenient center coordinates possible.   The "Auto Center" button is used to automatically center within the objects that are currently drawn.  Click on "Auto Center" and see the sliders move to their new locations.  To close the "Center" window, click on "Close" in the lower left corner.  Auto Center is also used to center the current image.

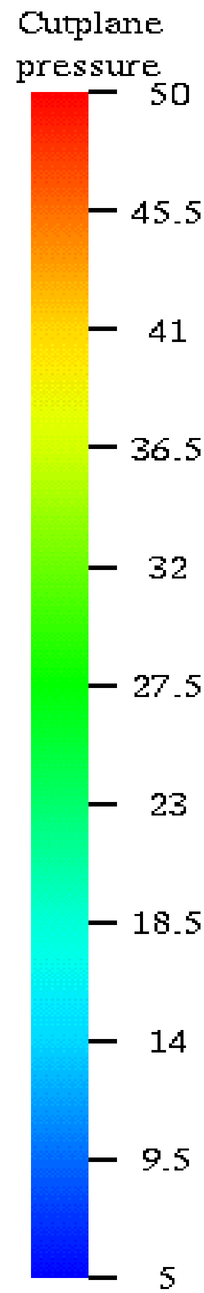



***Figure 5−9. Color bar***

## Color Bar:

The color bar option is in the Controls−1 menu.  When present, the color bar, (see Fig. 5−9), is a colored scale on the far left side of the main viewer.  It proceeds from blue at the low end to red on the high end and all the other colors of the spectrum in between.  There are numbered tick marks down the right side of the bar.  These marks tell you what each color represents numerically.  The color bar will be labeled on the top as to which drawing mode (nodes, cells, cutplane, tracers) and which field it currently represents, such as (i.e speed, pressure or temperature.)

**Turning on**

Click on the color bar option and a submenu will appear on the right.  Two basic operations can be performed in the submenu:  turning the color bar on or off, and stating a field preference.  Click on the "On"  box to toggle the color bar on or off.

**Preferences**

It is possible for several different color bar fields to be active simultaneously.  For example, you may want to superimpose the tracers on a color contour plot of speed along a cutplane.  GMV displays only one color bar at a time; therefore, GMV must decide which one to display.  This is the purpose of the color bar preferences option.  Underneath the on/off box is the word "Preferences" with a small triangle next to it.  Click on the word "Preferences."  A second submenu appears.  This submenu has five options in it: "None," "Nodes," "Tracers," "Cells," and "Cutplane."  The box next to the currently active preference will be highlighted. To select a preference, click on its corresponding box. The default preference is "None." When no preference is given, GMV uses a predetermined order of precedence. This precedence is given below:

**<highest>** 1) cutplane
2) cells
3) tracers
4) nodes

When a preference other than none is selected, GMV then makes the selection the highest priority.

## Color Edit:

The Color Edit option is located under "Controls−1" main menu control bar. To change default colors, GMV has a method for changing material and isosurface colors called "Color Edit." The color edit menu looks like Fig. 5−10.

### Changing the current color

The square in the center of the left side of the menu contains the current color. The current color can be changed in several ways. First, change it by using the red, green, and blue color slider bars located at the bottom left of the menu. Any color can be created using a combination of these

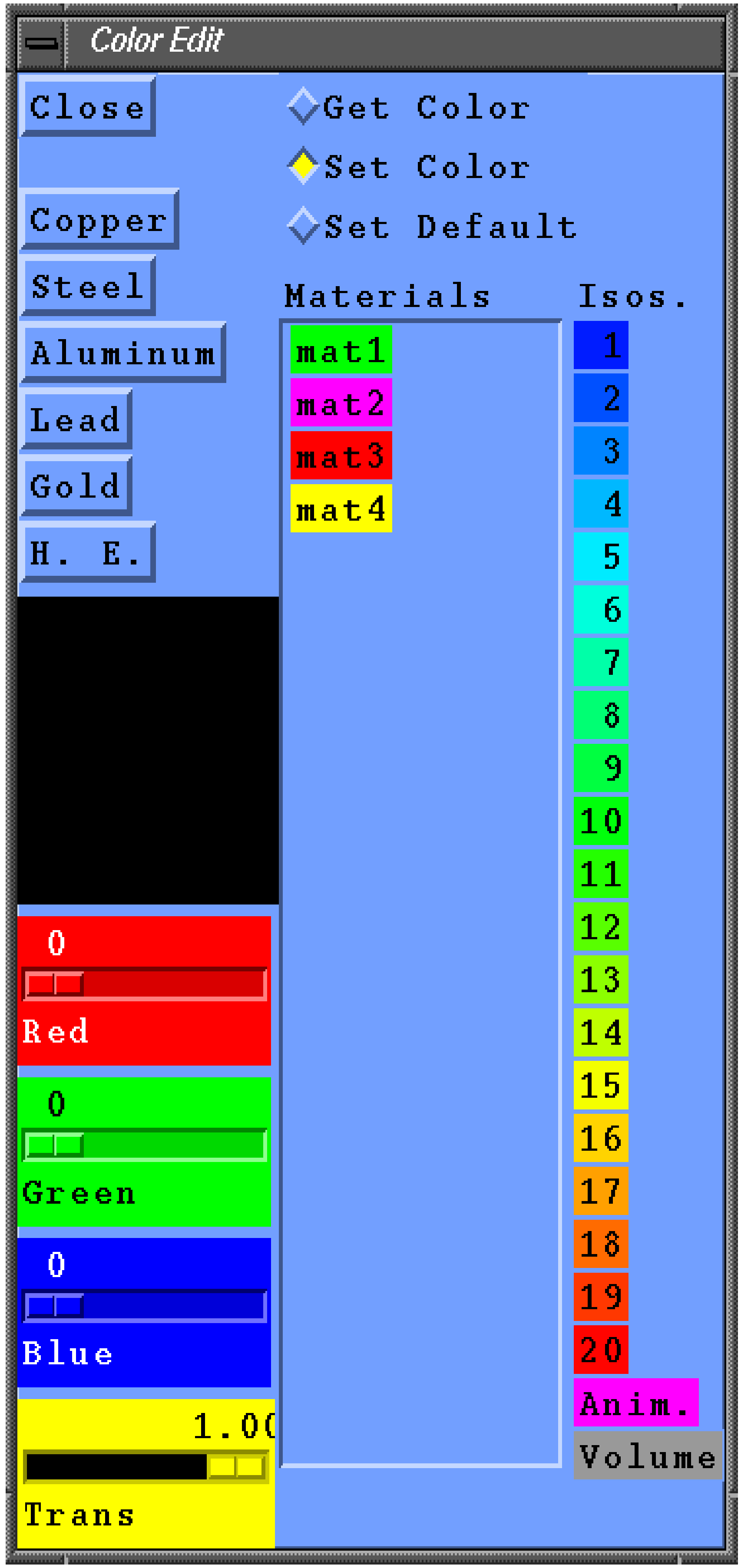



***Figure 5−10. Color Edit menu***

colors.  The current color’s transparency can also be changed with the transparency slider bar at the bottom of the menu.  A transparency value of one is no transparency, and a value of zero is  transparent.  Secondly, there are six predefined color buttons at the upper left of the menu labeled "Copper," "Steel," "Aluminum," "Lead," "Gold," and "H. E."  Clicking on any one of these changes the current color square to resemble those particular materials.  For example, the "Steel" button changes the current color to a grey, steel−like color.  The third way to change the current color is to click on the "Get Color" option in the upper right portion of the window.  When "Get Color" is selected, the current color will change to the color of the material or isosurface box clicked upon.  This is useful for copying colors from one place to another.

**Changing material or isosurface colors**

The right side of the "Color Edit" submenu is lined with material and isosurface color boxes.  The color of a particular material or isosurface box is the color that the material or isosurface will appear in, in the main viewer.  There is also one color box at the bottom of the list of isosurface colors labeled "Anim."  This is the color used for isosurface animation.  To change a material or isosurface color, first change the current color to the desired one.  Next, click on "Set Color" in the upper right part of the menu.  When "Set Color" is selected, any material or isosurface box you click on will change to the current color.  Therefore, any material or isosurface color may be reassigned.

At the bottom right of the "Color Edit" menu is a button to change the color of an isovolume.  This button works as described above; choose a color then click in the "Volume" button to set the color of the isovolume to the selected color.

**Reinstating default colors**

If you want to reset the GMV default colors, use the "Set Default" option.  When this option is highlighted, clicking on a material or isosurface box will return that box to its default color.

## Cycle:

"Cycle" is in the "Controls−1" menu.  Its function is to toggle on or off the cycle number in the upper−left corner of the main viewer.  Cycle numbers are sometimes included in input files to help you keep track of which simulation cycle generated the file.  If no cycle information exists in the input file, GMV will display a zero.

# The Controls−2 Menu

## Data limits:

The "Data Limits" option is in the "Controls−2" menu. This allows you to specify the minimum and maximum values of the color bar for Node Fields, Cell Fields, and for any tracers. When the limits of the color bars are changed, the object in the main viewer will appear in colors that correspond to the new range of the color bar. Setting new field limits is extremely useful when making time sequence movies. Set the limits to the simulation minimum and maximum field values to have a consistent color range throughout the time sequence.

When you click on the "Data Limits" option button in the menu, a submenu will appear on the right. From the sub menu, choose either "Node Fields," "Cell Fields," or "Tracers." After a selection is made, additional submenus will appear, depending on your choice.

**Fields**

Choosing "Node Fields" or "Cell Fields" will bring up a submenu similar to the one pictured here.(see Fig. 6−1) The current field whose limits may be changed is displayed after the "Field" label. To change the current field, click on the "New Field" button to pop up the node or cell field selection menu, depending on which submenu was selected. Below the current field are four boxes in which the original and current field limits for will be displayed. The original limits are the minimum and maximum value for the field as read from the input file. By default, the

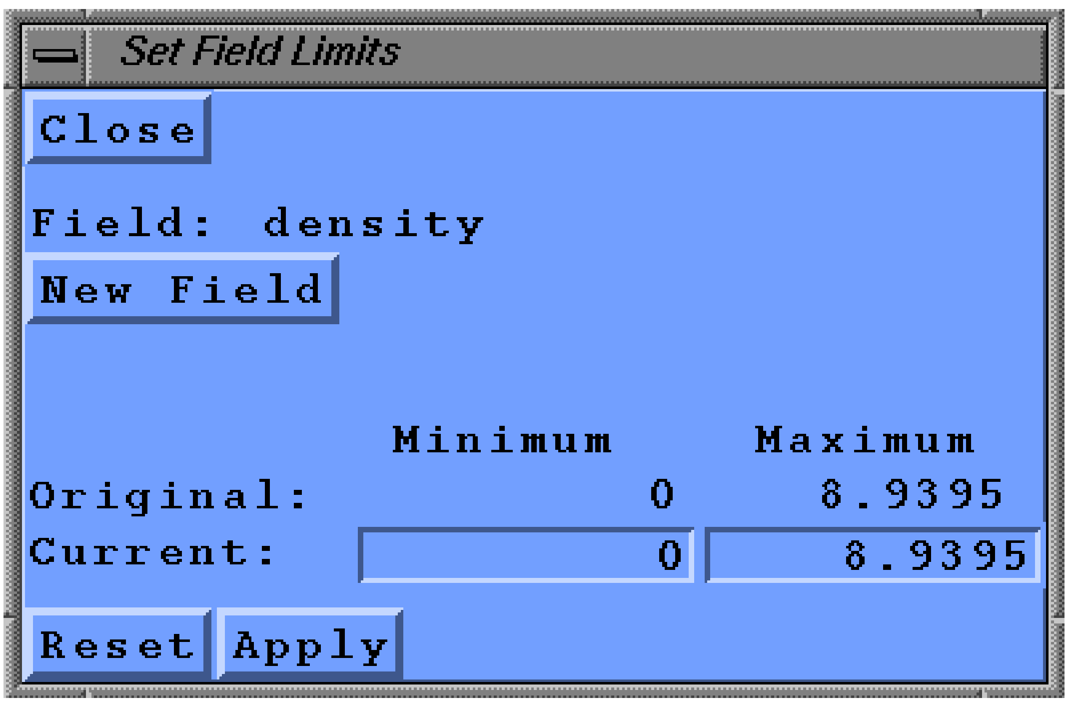



***Figure 6−1. Data Limits menu***

current limits are set to the original ones. This ensures that the spectrum of color covers all the available data evenly. To change the current values, click on the minimum or maximum current value box and enter the new limit. For the new limits to take effect, click on "Apply" in the lower−left corner of the submenu. To change the current limits back to the original ones, click on "Reset," and then apply the new limits. Click on "Close" in the upper−left corner of the submenu to exit the submenu.

**Tracers**

Choosing "Tracers" brings up a submenu labeled "Set Tracer Limits." The procedure for changing tracer limits is similar to changing node fields.

## Point Size: (OpenGL only)

The "Point Size" menu (Fig. 6−2) allows user specification of the size and shape of tracer points. This menu provides buttons used to select the rendering of tracer particles using two, four, six, or eight pixels. Additionally, the particle shape may be toggled to use either square or round particles.

The square particles update quickly, round particles rely on antialiasing to generate the particle curvature and thus update slower. Mesa OpenGL versions occasionally exhibit abnormal particle updates when antialiasing is used for particle generation.

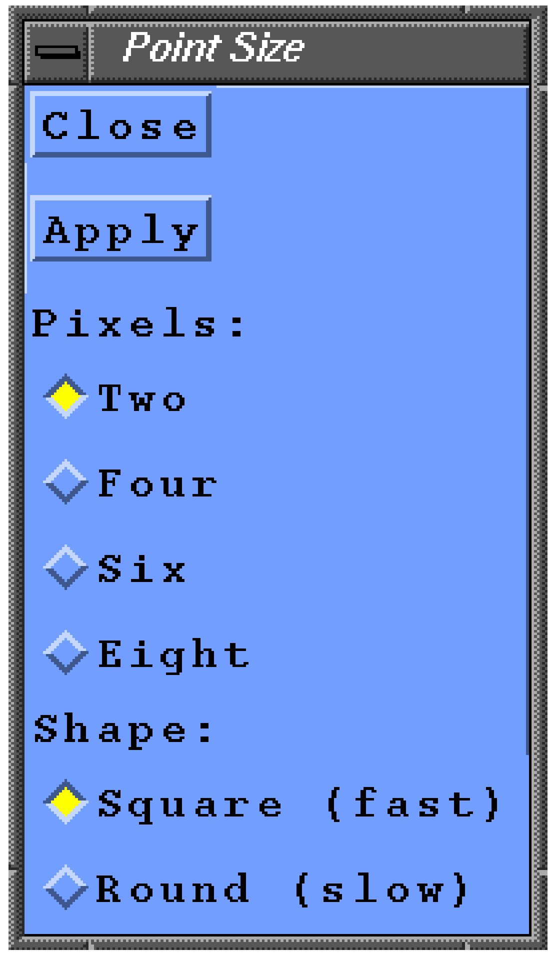



***Figure 6−2. Point SIze menu***

## Plot box:

The plot box is the three−dimensional plotting area. By default GMV chooses dimensions for the box to enclose all the data found in the input file in the smallest possible volume. To change the dimensions of the plot box, use the "Plot Box" option found in the "Controls−2" menu.

Click on the "Plot Box" button, to open the menu. A menu labeled "Set Plot Box" will appear (see Fig. 6−3). There are boxes to define minimum and maximum values on each axis such that any length, width, or height plot box can be created. To change a value, click on the appropriate box and enter a new

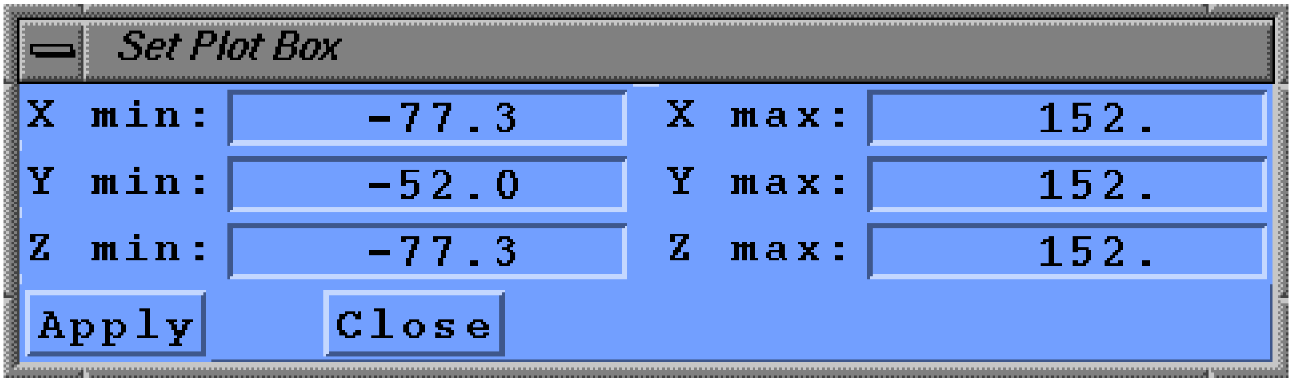



***Figure 6−3.  Plot Box menu***

number.  To cause the new box dimensions to take effect click on "Apply." Changing the dimensions of the plot box also affects the location of the center.  The center will automatically change to correspond to the center of the plotbox, but not necessarily the center of the object, which is default.   To return the center to the middle of the new plot box, open the center tool under "Controls−1" menu and click on "Auto Center" in the submenu.  Click on "Close" when modifications to the plotbox are complete.  Use plot box when generating time sequence animations with moving objects.  Set the plot box to the simulation minimum and maximum X, Y, and Z values so that the objects move across the screen.

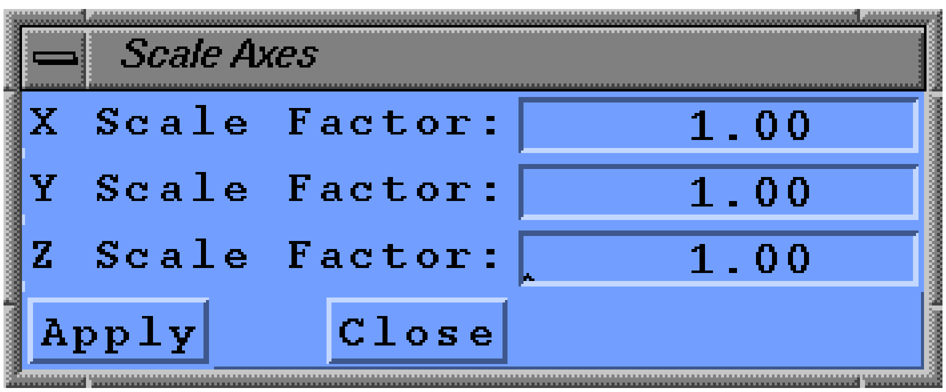



***Figure 6−4.  Scale Axes menu***

## Scale axes:

"Scale Axes" is under the "Control−2" menu.  Normally, all the axes in GMV are on an equal scale.   It may become necessary to exaggerate certain features of an object by stretching it in certain directions.  Changing the scale factors for the axes is done using the axes scaling tool.

To open the window, choose "Scale Axes" from the menu.  A submenu similar to Fig. 6−4 appears.  There are three boxes in the window, each containing a scale factor for a particular axis.  The first time the submenu is opened, you will notice that all the scale factors are set to one (default setting).  To change a scale factor for an axis, click on the box for that axis .  Enter the new scale factor and click on "Apply" button in the lower−left corner of the submenu.  GMV will redraw the object in the main viewer on the newly rescaled axes.  To close the window,

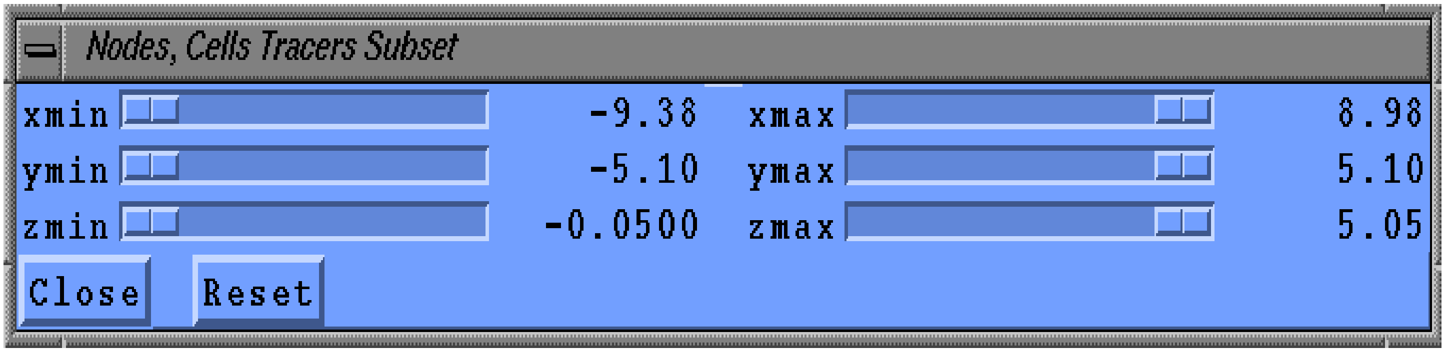



***Figure 6−5. Subset menu***

click on the "Close" button.

## Subset:

"Subset" is in the "Control−2" menu. A subset is a portion of all the available data. A subset box may be defined in GMV so that nodes, cells, tracers, or polygons within that box are shown. To define a subset, choose the "Subset" option from the menu. A submenu will appear on the right. Choose either "Nodes, Cells, and Tracers" or "Polygons," depending on the type of objects being manipulated.

**Nodes, cells, and tracers**

When this option is selected, a submenu similar to Fig. 6−5 appears. There are six slider bars: a minimum and maximum slider for each axis. The minimum and maximum values for each axis are used to create a box that houses the subset. To change the size of the subset box, drag the sliders back and forth until satisfied with the size of the subset box. All nodes and tracers outside the subset box will disappear, leaving only that data inside the box. Cells whose cell centers are outside the box will not be drawn. Click on the "Reset" button to reset the subset box. To close the window, click on "Close."

**Polygons**

Choosing the polygons option from the "Subset" submenu brings up the same polygon subset submenu explained on page 15.

## Time:

A GMV data file may contain a time index. Having the time index on the screen is useful for simulations. The "Time" button in the "Controls−2" menu is used to toggle the time display on and off. When the time is displayed it is in the upper right corner of the main viewer. If the input file contains no time data, GMV will display a zero.

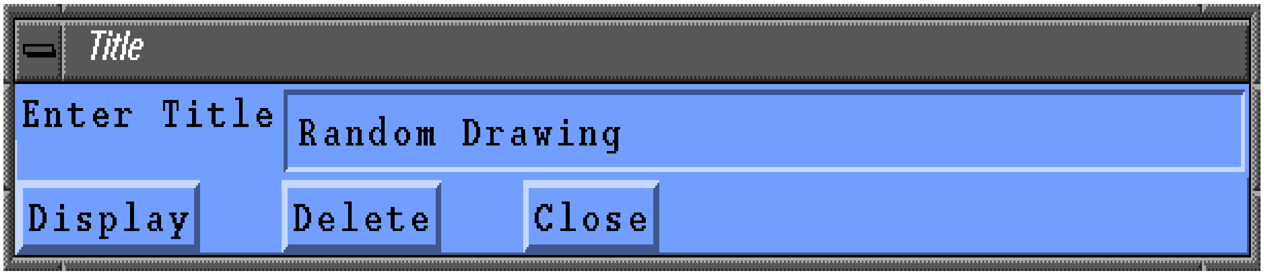



***Figure 6−6.  Title menu***

## Title:

A title may be added to the main main viewer.  To add a title, choose "Title" from the "Controls−2" menu.  A submenu  will appear (see Fig. 6−6).  Enter the text of the title into the box labeled "Enter Title."  Click on the "Display" button to add the title to the top center portion of the main viewer.  The submenu closes automatically.  To remove the title, reopen the submenu and click on the "Delete" button.

# *The Reflections Menu*

## X−axis Reflection:

Choose this button to reflect the mesh data about the X−axis.

## Y−axis Reflection:

Choose this button to reflect the mesh data about the Y−axis.

## Z−axis Reflection:

Choose this button to reflect the mesh data about the Z−axis.

## Mirror Imaging:

A special word of caution is in order.  Please note that reflecting about more than one axis is cumulative.  For example, GMV, when requested to reflect about an axis, will always reflect whatever is in the main viewer, even if half that data is reflected data.

Mirror imaging is useful for problems that have a great deal of symmetry in them.  The designer might want to include data for only half,  a fourth, or an eighth of the problem and then create the rest of the image by reflecting the data about an axis.  This method saves much time and memory when running the simulation.

# *The View Menu*

The options in the view menu are designed to allow three different ways to project the object on the main viewer. The three different ways GMV can display objects are: orthographic, perspective, and flight. Only one can be chosen at a time.

## Orthographic:

The orthographic projection is the default setting for GMV. It displays the object with parallel sides.

## Perspective:

The perspective projection represents object depth on the two−dimensional screen.

The difference between orthographic and perspective becomes clearer with an example: To draw a road going off into the distance, the orthographic drawing would be two parallel lines, whereas, perspective drawing would have two lines converging on a common point where the road is no longer visible (the vanishing point).

## Flight:

The "Flight" option is not really a special projection, rather a combination of the perspective view with special zooming−in capability that flies through the objects. When the flight mode is engaged, large red, green, and blue crosshairs will appear on the main viewer. These act as an aiming device. Mouse buttons operate differently in flight mode. The movements in flight mode simulate being in a helicopter. The left mouse button controls heading and pitch. Holding the left mouse button and moving left and right, changes the heading, where 0 degrees is parallel to the positive x axis. Moving up and down with the left mouse button pressed, changes the pitch angle where 90 degrees is level flight and 0 degrees is straight down. Holding the right mouse button and moving up and down, moves forward or backwards into and through objects. The middle mouse button still allows left, right, up, and down panning, with the capability of moving through objects. Thus interactive fly−throughs of GMV data from any direction are available.

# *The GMV Input Format*

## Input Specifications:

The format for GMV's input file follows. Please note that there are relatively few required entries, most data is optional, and keywords are used to identify its type. The data on the file can be either formatted ASCII or IEEE unformatted (but not both). Keywords are italicized and data names are in boldface. Example names for variables or flags are in double quotation marks. A description of the input line follows the data names or keywords. Only ***gmvinput***, the file type, node data, cell data, and ***endgmv*** are required, everything else is optional, however, each keyword may be used only once. For example, a second materials list is not allowed.

For IEEE unformatted files, keywords must be written as eight character words.

***gmvinput*** **file_type**

The first line identifies the file as a GMV input file with the file_type being either "ascii" or "ieee".

***nodes*** **nnodes −** Node points and number of points.

**x(nnodes)** − Float, x coordinates.
**y(nnodes)** − Float, y coordinates.
**z(nnodes)** − Float, z coordinates.

**OR** (for a structured regular brick mesh)

***nodes*** **−1 nxv nyv nzv** − Dimensions of structured regular brick mesh.

**x(nxv)** − Float, x coordinates.
**y(nyv)** − Float, y coordinates.
**z(nzv)** − Float, z coordinates.

**OR** (for a logically rectangular brick mesh)

***nodes −2*** **nxv nyv nzv**
**x(nxv * nyv * nzv)** − Float, x coordinates.
**y(nxv * nyv * nzv) −** Float, y coordinates.
**z(nxv * nyv * nzv)** − Float, z coordinates.

**OR** (to read node data from a separate file)

***nodes fromfile "*filename*"*** **−** Read node data from *filename.*

***cells* ncells −** Cell data and number of cells.

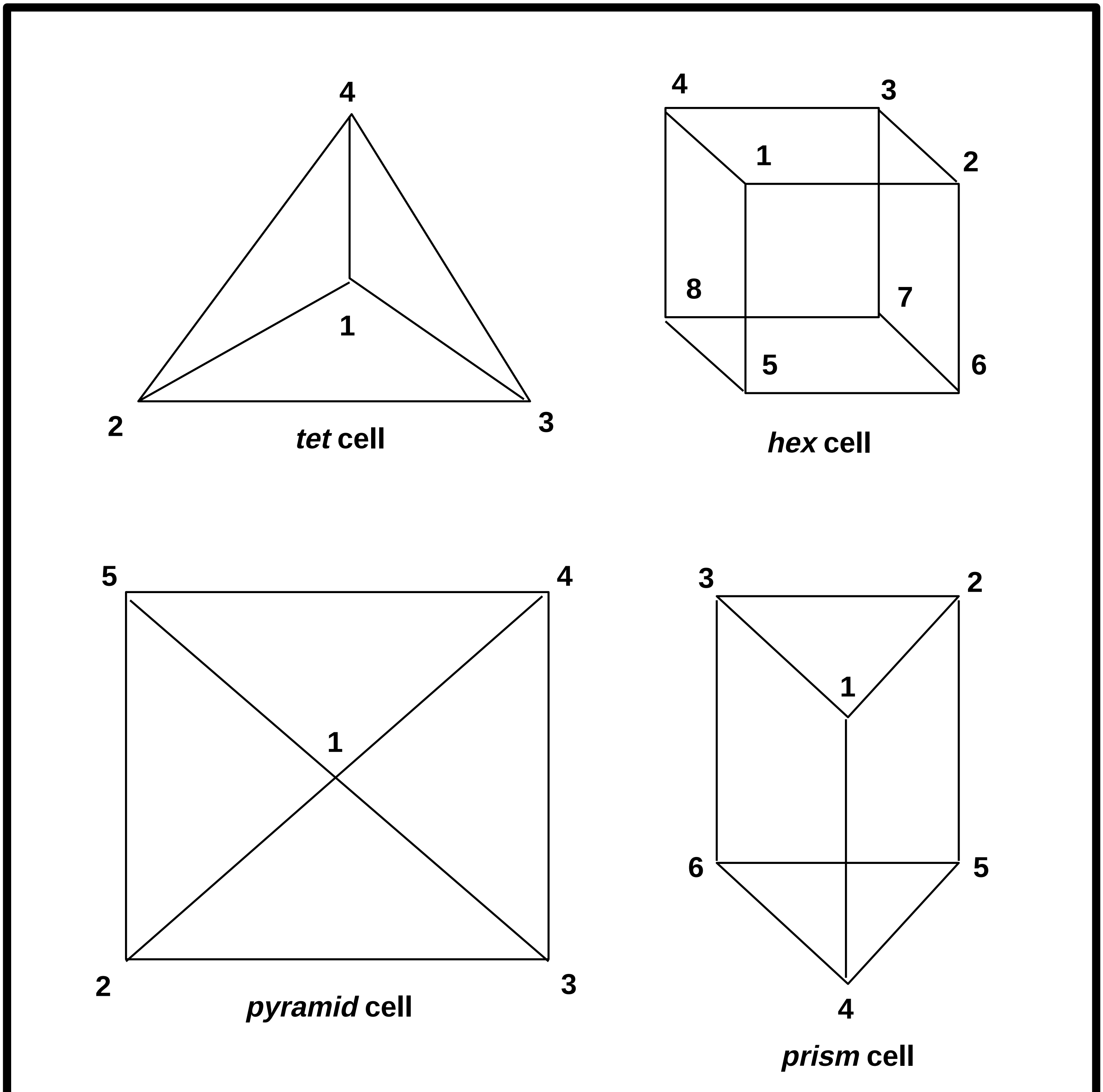



***Figure 9−1. Cell vertex order***

**OR** (to read cell data from a separate file)

***cells fromfile "*filename*"*** − Read cell data from *filename.*

**cell_type number of elements** − Cell data; format depends on cell type, vertex or face data. See examples.

Examples of cell types (Note: cell types can be mixed):

General type, can be used to define any cell volume.

***general* nfaces** − General type, number of faces in cell.
**nverts(nfaces)** − Number of vertices per face.
**vertex_ids(sum(nverts))** − Integer list of node numbers that define the polygonal faces.

***tri* 3** − Triangular cell, with 3 vertex ids.
**verts(3**) − vertex ids.

***quad* 4** − Quadrilateral cell, with 4 vertex ids.
**verts(4**) − vertex ids.

***tet* 4** − Tetrahedral cell, with 4 vertex ids.
**verts(4**) − vertex ids.

***hex* 8** − hexahedral cell, with 8 vertex ids.
**verts(8)** − vertex ids.

***prism* 6** − Prism cell, with 6 vertex ids.
**verts(6)** − vertex ids.

***pyramid* 5** − Pyramid cell with 5 vertex ids.
**verts(5)** − vertex ids.

Note: the ordering of the vertices for tet, hex, prism and pyramid is shown in Fig. 9−1. The tri and quad cells are two−dimensional entities that employ a sequential vertex numbering scheme around the periphery of the cell.

***material*** **nmats data_type** − Material data, number of materials, and data type (0=cells, 1=nodes).

**matnames(nmats)** − 8 character material names.
**matids(ncells)** − Integer, material ids for cells.
or
**matids(nnodes)** − Integer, material ids for nodes.

**OR** (to read material data from a separate file)

***material fromfile "filename"*** − Read material data from *filename.*

***velocity*** **data_type** − Velocity data, and data type (0=cells, 1=nodes)

**u(ncells)** |
**v(ncells)** | For cells.
**w(ncells)** |
or
**u(nnodes)** |
**v(nnodes)** | For nodes.
**w(nnodes)** |

***variable*** − Keyword indicating that other cell or node data sets follow. The data sets have the form:

**"anyname" data_type** − An eight character name for the data, the data type (0=cells, 1=nodes)
**data(ncells or nnodes)** − array of float data.

Examples (<u>DO NOT</u> use quotes in actual file):
"density" 0
density_data(ncells)

"temp" 0
temp_data(ncells)

"pressure" 1
pressure_data(nnodes)

***endvars*** − Keyword indicating end of variable data.

***flags*** − Keyword indicating that selection flag data sets follow.

**OR** (to read flag data from a separate file)

***flags fromfile "filename"*** − Read flag data from *filename.*

The data sets have the form:

**"anyname" ntypes data_type** − Flag name, number of flag types, and data type (0=cells, 1=nodes).

**flagnames(ntypes)** − 8 character flag type names.

**iflag(ncells)** − Integer, flag ids for cells.

or

**iflag(nnodes)** − Integer, flag ids for nodes.

Examples:
"nodetype" 4 1
"inactive" "interior" "interfac" "boundary"
node_data(nnodes)

"cnstrain" 3 0
"static" "piston" "air"
cnst_data(ncells)

***endflag*** − Keyword indicating end of flag data.

***polygons*** − Keyword indicating surface polygon data follows. (eg. interface or boundary faces.)

**material_no nverts x(nverts) y(nverts) z(nverts)**

**OR** (to read polygon data from a separate file)

***polygons fromfile "filename"*** − Read polygon data from *filename.*

Where:

**material_no** − Integer number related to material data.

**nverts** − No. of vertices.

**x(nverts)** − x coordinate of polygon vertices.

**y(nverts)** − y coordinate of polygon vertices.

**z(nverts)** − z coordinate of polygon vertices.

This data is repeated for all polygons.

***endpoly*** − Keyword to indicate end of polygon data.

***tracers* ntracers** − Tracer points and the number of tracers input.
  **x(ntracers)** − Float, x coordinates.
  **y(ntracers)** − Float, y coordinates.
  **z(ntracers)** − Float, z coordinates.
Followed by trace data of the form:
  **"anyname"** − An eight character name for the data.
  **data(ntracers)** − array of float data.

Examples:
"temp"
temp_data(ntracers)

"pressure"
pressure_data(ntracers)

***endtrace*** − Keyword indicating end of variable data.

***probtime* ptime** − Keyword and floating point problem time value.

***cycleno* cycleno** − Keyword and integer problem cycle number.

***endgmv*** − Keyword signifying the end of the input file.

## Input Data Details:

**Header**

The header line contains the "gmvinput" keyword and the character variable file_type which contains either "ascii" or "ieee". The ASCII file type indicates that the file was written as a formatted ASCII file, the file will be read using list−directed I/O so there must be at least one space between data elements. The IEEE file type indicates the file was written as an unformatted file with IEEE single−precision floating point, character and 32−bit integers. When writing IEEE binary files, be sure that all keywords and character data must be eight characters long.

**Nodes**

The "node" keyword describes the beginning of cell node data points and the variable nnodes on this line are the number of nodes (ie. the length of the node data arrays that follow). The next three lines are the three floating point arrays that represent the X, Y, and Z coordinates of the nodes.

The ***nodes*** keyword has three alternate forms. The first is used to generate a structured, regular brick mesh. Entering **−1** for the number of nodes signifies this alternate syntax. After **−1** on the same line are the dimensions of the mesh; first the number of nodes along the X−axis, then the number along Y, and the number along the Z−axis. The three lines that follow contain the X, Y, and Z coordinates of the nodes along each axis, which will be used by GMV to generate the entire mesh. Note: because GMV uses this information to generate a large mesh of cells, the number of cells specified with the ***cells*** keyword must be zero.

The second alternate syntax for the ***nodes*** keyword is used to generate a logically rectangular structured mesh. Entering −2 for the number on nodes signifies this alternate syntax. After −2 and on the same line are the dimensions of the mesh; first the number of nodes along the X−axis, then the number along Y, and the number along the Z−axis. The three lines that follow contain the X,Y, and Z coordinates of the nodes for all nodes (nx*ny*nz), which will be used by GMV to generate the entire mesh. Note: because GMV uses this information to generate a large mesh of cells, the number of cells specified with the ***cells*** keyword must be zero. For any meshes that are closed you need to repeat the necessary nodes for closure.

The third form of the ***nodes*** keyword is used to specify the existence of node data in an external file:

***nodes fromfile "*****filename"**

This syntax is used within the scope of the main GMV file to instruct GMV that node data is located in a *fromfile* specified by "**filename"**. **Filename** is a user supplied character string that must be enclosed by double quotes. The use of this keyword form specifies that the fromfile will contain the pertinent data in the same format and context as would be used in the main GMV file; the fromfile must be a valid GMV format file. When GMV encounters this keyword form, main file processing stops and the fromfile is opened and searched for the applicable data. Once the data is input, the fromfile is closed, and main GMV file processing continues.

Fromfiles are useful within GMV for displaying animation sequences and the production of movies. In many animation sequences, much of the data remains unchanged between frames (for example, nodes, cells, material, flags, and polygon data may remain constant if the problem domain and physical geometry of the problem does not change between frames). Constructing distinct and complete GMV files of each frame consumes much disk space needlessly; the fromfile capability allows the placement of constant data (i.e., one or more of node, cell, material, flag, and polygon data) within a single file that will be repeatedly be referenced by several GMV input files. This constant file is tagged a fromfile in this implementation.

**Cells**

The ***cells*** keyword indicates the beginning of cell descriptions. The variable ncells on this line are the number of cell descriptors that follow. There are six standard cell types that GMV can read, ***tri, quad, tet, hex, pyramid*** and ***prism***. The type is followed by the number of vertices contained in the cell, 3 for tris, 4 for quads, 4 for tets, 8 for hex, 5 for pyramid, and 6 for prisms. The next line contains the node numbers for the cell vertices. The vertex ordering for selected standard cell types is shown in Fig. 9−1. The tri and quad cells are simple two−dimensional entities that employ a sequential vertex numbering scheme counterclockwise around the periphery of the cell.

The ***general*** cell type is available for nonstandard cells. These cells are described by their faces. The nfaces variable indicates the number of faces for the cell. The next line of data is the number of vertices for each cell face. The third line of the set contains the node numbers of the vertices for each face for all faces. The integer array size for the nodes will be the sum of the vertices for the cell faces. The faces do not have to be specified in any order. However, the vertices for each face must be specified in an order that describes the face polygon.

The node and cell data are required and must be in order, although the number of cells can be zero if no cells exist. Note, there is no external numbering for the nodes and cells; the order of input is the numbering sequence for both nodes and cells.

Cell data input may also be performed by consulting a remote file (see the ***fromfile*** description in the ***nodes*** section, above). For this option, the syntax and function is identical to the ***fromfile*** keyword described in the ***nodes*** section.

**Materials**

The keyword ***material***, denoting material data, is an optional but highly recommended input data type. Up to 128 materials are allowed. On the keyword line are the variables nmats (1 to 128), the number of materials available (1 to nmats), and data_type ( 0 means the material data is cell centered and 1 means the material data is node centered). The next line of data is the eight character names given to the nmats materials. Finally, the last line of material data is the cell or node centered material ids; this is an integer array. Material data is necessary if surface polygons exist.

Material data can be used to distinguish between different classes of cell or node data besides the normal engineering material definitions. For example, the material data can be density layers for an ocean model, horizons in seismic data, or rock layers in a reservoir model.

Material data input may also be performed by consulting a remote file (see the ***fromfile*** description in the ***nodes*** section, above). For this option, the syntax and function is identical to the ***fromfile*** keyword described previously.

**Velocities**

The keyword ***velocity*** indicates that optional velocity data follows. Again the **data_type** value of 0 indicates cell−centered velocities and a value of 1 indicates node−centered velocities. The next three lines of data are the u (x component), v (y component) and w (z component) velocity floating point arrays. Cell−centered velocities will be averaged and saved as node−centered velocities. Also, speed and kinetic energy variable fields will be automatically generated and added to the end of the input variable fields.

**Variable data fields**

The ***variable*** keyword is used to denote the beginning of any other cell or node data fields. The data are entered as a group for each field variable. Up to 100 different field variables are allowed and each field variable is named by the user. The ***endvars*** keyword is used to end the field data input. Each field data variable is defined by two input lines. The first line contains the eight character name of the variable and the data_type of the field (0−cell data, 1−node data). The second line is the floating point array for the cell or node data. Cell−centered field data will be averaged and stored as node−centered data.

**Selection Flags**

The ***flags*** keyword means that integer selection flag data sets follow. Up to 10 different types of selection flags and up to 128 different flag values per flag are allowed. These data sets can be any type of integer data that can be used to select a node or cell for display purposes. The names for the flags and for the flag types are placed in selection buttons in a menu. The integer data must be a number between 1 and ntypes (1 to 128). The ***endflag*** string ends the flag data.

Flag data input may also be performed by consulting a remote file (see the ***fromfile*** description in the ***nodes*** section, above). For this option, the syntax and function is identical to the ***fromfile*** keyword described previously.

**Polygons**

The ***polygons*** keyword indicates that surface polygons data follows. The surface polygons can be interface or boundary polygons for a material. Each line describes one polygon. The line contains the material number (1 to **nmats**) associated with the polygon, the number of vertices in the polygon and the x, y, z arrays that define the vertices for the polygon. The ***endpoly*** string terminates the polygon data.

The polygons keyword can be used to describe any surfaces a simulation can generate. Be sure to give each surface a material number and that this material number has a material name listed under the ***materials*** keyword.

Polygon data input may also be performed by consulting a remote file (see the ***fromfile*** description in the ***nodes*** section, above). For this option, the syntax and function is identical to the ***fromfile*** keyword described prevously.

**Tracers**

The ***tracers*** keyword indicates that tracer particle data (or any point data other than node data) follow. The ntracers variable following the keyword is the number of tracers that are input. The next three lines are the x, y, and z floating point coordinates of the tracers. Following the coordinates are the variable data fields for the tracers. The data is entered as a group for each field variable. Up to 40 different field variables are allowed. Each tracer field data variable contains an eight character variable name followed by a floating point data array. The ***endtrace*** string terminates the tracer data.

**Problem Time**

The ***probtime*** keyword is followed by a floating point number that represents the simulation problem time. This value is displayed at the top right corner of the main viewer.

**Cycle Number**

The ***cycleno*** keyword is followed by an integer number that represents the familiar cycle number. This value is displayed at the top left corner of the main viewer.

## Sample input data:

The following is a sample GMV input file in ASCII format. It includes most of the features and commands GMV allows. When read in, the file creates a cube with several other different cell types attached to it. The additional cells are: one tetrahedral cell, one prism cell, one pyramid cell, and one general cell. The general cell has ten faces, and could be described as an octagonal prism. There is a large cube constructed from square polygons, each with a different material, that encloses all of the cells. In addition, variables, tracers, and flags with arbitrary data have been included so that you may see the format for entering such elements into an input file. The keywords are italicized. In addition, blank lines have been inserted between major elements of the file for clarity, but these are not necessary in a real input file. The input data follows:

***gmvinput*** ascii

***nodes*** 28

0 50 50 0 0 0 50 50 25 25 25 25 50 50 50 50 50 50 50 50 80 80 80 80 80 80 80 80
0 0 0 0 50 50 50 50 75 0 50 −25 16.7 33.4 50 50 33.4 16.7 0 0 16.7 33.4 50 50 33.4 16.7 0 0
50 50 0 0 50 0 0 50 25 80 80 65 0 0 16.7 33.4 50 50 33.4 16.7 0 0 16.7 33.4 50 50 33.4 16.7

***cells*** 5

***hex*** 8
1 2 3 4 5 8 7 6

***pyramid*** 5
9 5 6 7 8

***prism*** 6
10 1 2 11 5 8

***tet*** 4
12 1 2 10

***general*** 10
8 8 4 4 4 4 4 4 4 4
13 14 15 16 17 18 19 20 21 22 23 24 25 26 27 28 19 27 28 20 20 28 21 13 13 21 22 14 14 22 23 15 23 24 16 15 24 25 17 16 25 17 18 26 26 18 19 27

***material*** 6 0
mat1
mat2
mat3
mat4
mat5
mat6
1 2 3 4 5

***polygons***
1 4 −100 100 100 −100
100 100 −100 −100
100 100 100 100
2 4 −100 −100 100 100
−100 100 100 −100
−100 −100 −100 −100

```
3 4 −100 100  100 −100
    100 100  100  100
    100 100 −100 −100
4 4 −100  100  100 −100
   −100 −100 −100 −100
    100  100 −100 −100
5 4  100 100  100  100
   −100 100  100 −100
    100 100 −100 −100
6 4 −100 −100 −100 −100
   −100 −100  100  100
    100 −100 −100  100
```

***endpoly***

***tracers*** 10
0 20 40 60 80 100 120 140 160 180
0 20 40 60 80 100 120 140 160 180
0 20 40 60 80 100 120 140 160 180
pressure
0 5 10 15 20 25 30 35 40 45

temp
45 40 35 30 25 20 15 10 5 0

density
0 5 10 15 20 25 30 35 40 45

***endtrace***

***velocity*** 0
0 0 5  0  0
0 5 5  0 10
5 5 5 10 10

***variable***

pressure 0
0 5 10 15 20

temp 0
0 5 10 15 20

density 0
0 5 10 15 20

***endvars***

***flags***

flagtype 3 0
good bad ugly
1 2 3 2 1

stufftype 3 0
bing bang boom
1 2 3 2 1

***endflag***

***endgmv***

# *Making Movies With GMV*

## GMV command line options:

GMV has several command line options which can be used to create time sequence movies of mesh data, noninteractively. In this manner, GMV can be included in shell scripts that create snapshots. The following are the available GMV command line options:

window size: ***−w xloc yloc width height***

**xloc, yloc:** X and Y coordinates of lower left corner of window where GMV will draw the object contained in the input file.
**width, height:** the width and height in pixels where GMV will draw the object contained in the input file.

movie mode (noninteractive): ***−m***

Starts GMV noninteractively solely for the purpose of creating snapshots; therefore no main GMV window will appear. The only indication that GMV is actually doing something will be GMV's usual status messages printed in the shell.

input file name: ***−i filename***

Specifies an input file for GMV to read and is used only with the **−m** option.

attributes file name: ***−a attribute_filename***

Specifies an attributes file for GMV to use to draw the mesh data and is used only with the **−m** option.

snapshot: ***−s***

Takes a snapshot of the object in the input file and by default saves it as "AzsnapgmvAz" in SGI RGB format and is used only with the **−m** option.

Here is an example script that uses GMV's command line options to create a series of snapshots for use in movie making:

```
#
set verbose
set k=(1−21)
foreach i (k)
/usr/local/bin/gmv −m −a iso.attr −w 639 0 640 512 −i gmvout3.$i −s
mv AzsnapgmvAz ksbmiso.$i.rgb
end
```

Of course, you could just use the movie utility described below instead.

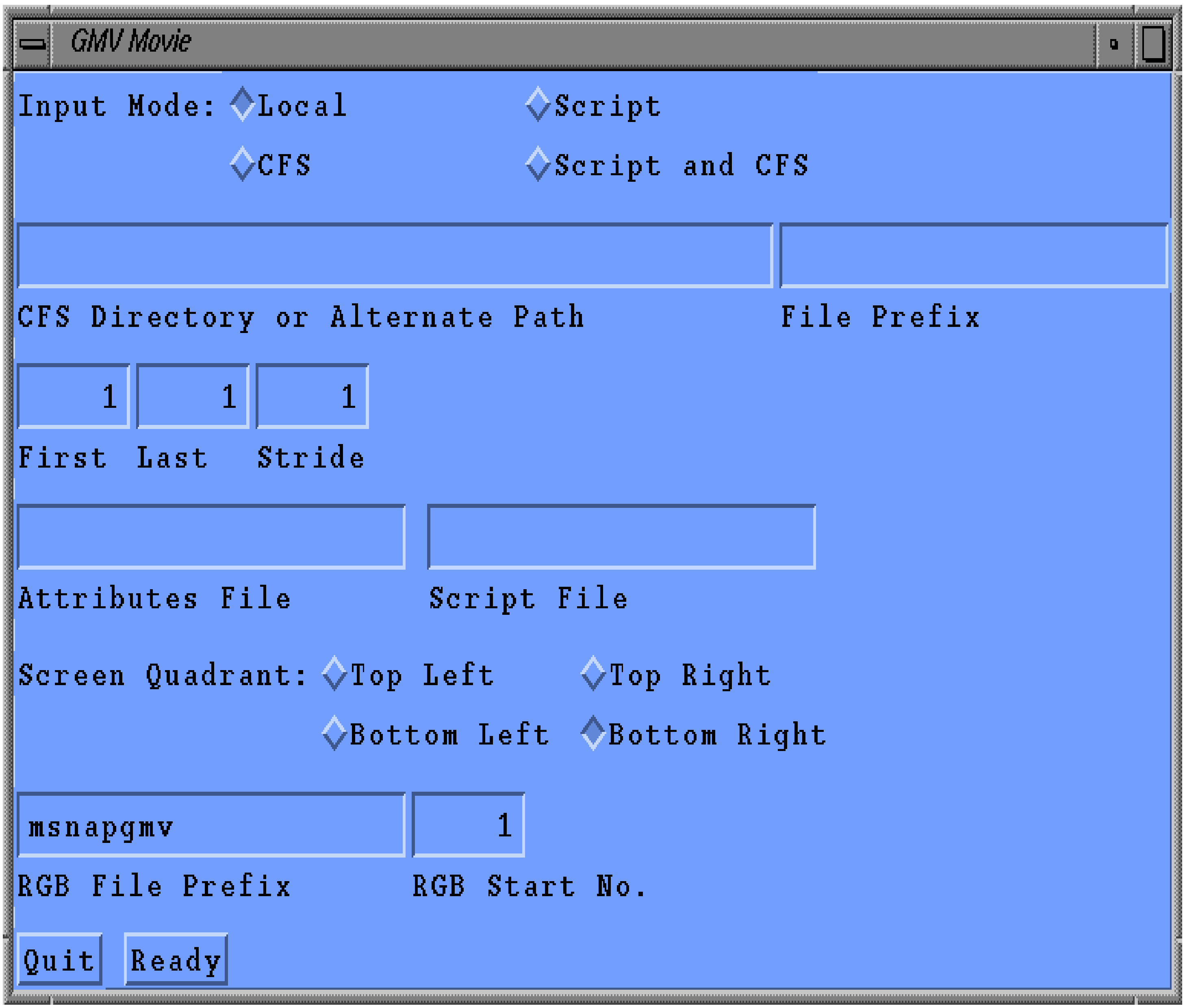



***Figure 10−1. GMV Movie menu***

## The GMV Movie utility:

The GMVMOVIE utility can be used instead of a script to generate time−sequence movies, (see Fig 10−1). It works by first locating where the files are, then getting them one at a time and running gmv to make a snapshot image file of each frame. As GMVMOVIE finishes each frame, it removes the input file if the file was imported from CFS. If the input file was on a local disk, GMVMOVIE leaves it there.

After running GMVMOVIE, you may use the SGI utility "movie" to look at the snapshot frames interactively. The source for "movie" is provided in 4DGifts. The usual path name is ~4DGifts/iristools/imgtools.

Here are the steps for making a movie with GMVMOVIE:

**1.** Generate a series of gmv input files. These files must have the following naming scheme:

prefix + sequence,

where prefix is a string common to all the files (e.g.: gmva) and sequence is a 3−digit sequence number. For example, in the filename gmve023, the prefix is "gmve" and the sequence is 023. These files must reside on the Common File System (Los Alamos National Laboratory's mass storage system) or on a local disk.

**2.** Move one of these GMV input files to the workstation, by using CFS and run gmv. The reason for doing this is to adjust the image from this file display relevant information for the attributes file. This will determine the viewing angles, magnification, visible materials, fields, cutplane parameters, and contour surfaces that will be applied to each frame of the movie. A helpful hint is to use a file late in the time sequence to set up attributes. An important point is that the range of values for any fields in this GMV file will subsequently apply to all frames of the movie. This makes the color−coding for cutplanes or bounding box fields consistent from frame to frame. Any field value in other files that is below the range in this file will be colored the minimum field color (dark blue); values above the range will be colored the maximum color (bright red).

When setup is complete, click on the "Put Attributes" button and specify a name for this attributes file. (Hint: use the suffix ".attr" to differentiate attributes files from other files.)

**3.** Start up GMVMOVIE and fill in the blanks on the menu, (see Fig. 10−1).

The Input Mode options are "Local", "CFS", "Script", "Script and CFS". "Local" means that the GMV input files are stored on the workstation, usually in the current directory.  "CFS" means the files are stored on CFS (LANL's Common File System mass storage system).  "Script" means the files are stored locally, but the file prefix and the first, last, and stride values are read from the script file.  This is useful in combining GMV input files with different file prefix names into one movie. "Script and CFS" is used when the files are stored on CFS and a script file determines the GMV input files to retrieve from CFS.

The box labeled "CFS Directory or Alternate Path" should contain the CFS path or Unix path (if different from the current directory).  If the data is in the current directory, leave this box blank.  Enter the prefix of the file names in the "File Prefix" box.  For example, if the input files are stored on CFS node /my/mesa/node and the files are named gmvc001, ..., gmvc044, then the "CFS Directory or Alternate Path" box should contain:

/my/mesa/node

and the "File Prefix" box should contain:

gmvc

If you select "CFS" as the input mode, then GMVMOVIE will prompt you for a CFS password after you click on "Ready".  If the CFS node has no password, press the Enter key.  If a password is required, type it in after the prompt.  It will not print on the screen.

Fill the "First", "Last", "Stride" boxes with the beginning, end and interval of the GMV input files.  For example, entering 1, 50, 5 with a prefix of gmvin1 will create movie frames for every fifth file from gmvin1001 through gmvin1050.

The "Attributes File" box should contain the attributes file name saved with the "Put Attributes" file option.

The "Script File" box is used when "Script" or "Script and CFS" input mode is selected.  Enter the file name of the script file.  For an example of a script that may be used here, please look under **Other Useful Information** below.

There are four options in the "Screen Quadrant" selection.  Be sure that no other graphics window is running in the quadrant selected.  The window that is drawn is 640 by 512 pixels; if the workstation does not have a 1280 by 1024 pixel resolution, then always select the upper left quadrant.

The "RGB File Prefix" box is used to enter the file name prefix of the snapshot files.  The default name is "msnapgmv".  This can be changed by the user.  The "RGB Start No." contains the starting sequence number for the snapshot files.  For example, if the "RGB File Prefix" contains rgbout and the "RGB Start No." is 50, then the snapshot file names will start with rbgout050.  This is extremely helpful when a sequence has been interrupted and a restart is necessary.

Click on the "Ready" button to start the movie process.

**4.** After you click on "Ready", GMVMOVIE gets the first file and starts up GMV. If everything is correct, GMV will make an image in the selected quadrant of the screen, and then GMVMOVIE will save it in a file with the rgb file prefix and a three digit sequence number. You may do other work on the workstation while GMVMOVIE is running, but be careful not to obscure the selected quadrant of the screen. If this happens, whatever is put there might be captured in one of the snapshot image files.

**5.** The GMVMOVIE utility will output information in the executing window as it proceeds through its process to let you know what it's doing.

**6.** After GMVMOVIE finishes, you may look at the snapshot files it created by running the SGI utility "movie." To run movie, type:

```
movie (rgb file prefix)*
```

If you type:

```
movie msnapgmv* −z 2
```

it will fill the entire screen.

**Other useful information**

If you do not enter enough information, then when you select "Ready", the menu will remain and an error message appears. You must select one input mode and have the input file prefix and the attributes file name entered if input is local or from CFS. If the input mode is Script or Script and CFS the file prefix and the start, end, and stride data are read from the script file. The Script input mode can be used to combine GMV input files with different prefixes. For example, the following script data will process gmvfiles gmva001 to gmva005 then files gmvb001 to gmvb010:

```
gmva 1 5 1
gmvb 1 10 1
```

If you have GMV input files in your directory and they have the same names as files that GMVMOVIE will be getting from CFS, the existing input files will be overwritten.

You can run up to four gmvmovies at a time by selecting different quadrants for each movie, as long as the workstation has a 1280 x 1024 pixel resolution. Just make sure that the quadrants do not overlap and there are no other graphics images or menus running in the quadrants.

If you cannot get GMVMOVIE to stop, go to another window and type

```
ps −ef | grep gmvmovie
```

You will see something like this:

```
jxf  5503  4991 80 11:35:36 ttyq9     0:45 gmvmovie
jxf  5507  5471  1 11:36:28 console   0:00 grep gmvmovie
```

Locate the process number for GMVMOVIE (in this case, 5503) and then kill it by typing

kill (process number), e.g.: kill −9 5503.

If you are running the OpenGL version of GMV on an SGI, the window manager will require you to place the window on the screen, contrary to the −w option. To prevent this interactive placement problem, add the following lines to your .Xdefaults file, then log in to activate the resources.

```
4DWm*interacitvePlacement:false
4DWm*GMV*clientDecoration:none
```

# Helpful Hints

1) Nodes are more visible with a black background. To change the background color, use the background color controls near the top of the main GMV window. All three colors (red, green, and blue) must be all the way to the left for a black background.

2) Shaded objects such as polygons and isosurfaces look better with a white background. All three color controls must be to the right for a white background.

3) When selecting polygons, turn off the "Shaded" and "Edges" buttons. First select everything you need to view, and *then* shade the polygons. This way, GMV only has to update the screen once, instead of every time you change your selection. This is especially helpful for large problems.

4) Hint number three also applies for cells. Turn everything off and select what you need first. Then turn on faces, or edges as needed.

5) Workstations without any graphics acceleration or ones running the Mesa (OpenGL to X) version are especially slow running GMV. Therefore, it is to your advantage not to overwork the machine and only force GMV to do something when it is necessary. The mouse controls are not very useful when running the slow version of GMV. It is more efficient to use the slider bars above the main viewer to manipulate the object. This will be more accurate and save time. Starting GMV with a smaller window will also speed up the interactive drawing process. For example, you could invoke GMV with the **−w** option like this:

**gmv −w 0 0 500 400**

which will start GMV with a 500x400 pixel main viewer instead of the usual 900x700 pixel main viewer. Setting the interactivity slider bar to its highest setting is helpful. Thus, when you manipulate the object in the main viewer, less data will display until manipulations are complete. To guide you during this process, turn on the bounding box as a reference tool. It will not disappear during the interactive drawing process as the data does.

6) If you open GMV with a window smaller than 640x512, do not attempt to save movie size snapshot images. These snapshots will be incorrect.

7) There are many image conversion utilities available that can convert SGI RGB to other image formats. One is called ***imtools*** and is available at ***ftp.sdsc.edu*** (the San Diego Super Computing Center). The compressed tape archive of these utilities is about 11 megabytes. The relevant tool included in this package is called ***imconv***. This tool can convert files into practically any format from any format.

Another helpful tool for image conversion is xv. The URL for this tool is: ***http://www.sun.com/sunsogt/catlink/xv/xv.html***.

8) Material isosurfaces: GMV will generate material isosurfaces using node values. There must be a change in material values in a cell before isosurface elements can be generated for the cell. This isosurfacing method causes a problem for simulations where boundary or interface materials lie along cell faces. Isosurface elements will not be generated along these cell faces. Thus, it is much better for you to generate your own material surfaces and enter them in the ***polygons*** list.

9) Null material number: Nodes or cells with a 0 (zero) material number are not drawn and are not included when calculating field minimums and maximums. A "zero" material number can be useful if your simulation deletes nodes but keeps a continuous numbering sequence. You can keep deleted nodes so you do not have to renumber cell vertices; just enter a 0 (zero) material number for those nodes and GMV will ignore them unless requested.

10) If polygons are the only necessary elements of your simulation, you must still enter data for nodes, cells, and materials. There may be zero cells, but there must be at least one node. Place this node somewhere within the polygon boundaries. In the materials list, the single node must be assigned a material number. An example GMV input file follows:

***gmvinput*** **ascii**
***nodes*** 1
0
0
0
***cells 0***
***material*** 3 1
mat1

mat2
mat3
1
***polygons***
{ polygon data }
***endpoly***
***endgmv***

10) Menu fonts: You can use standard X resource files to change the font or font size of the menus. Using a smaller font is useful when you star GMV with a small drawing window using the −w option. An example of the resource line for a 14 point font follows:

GMV*fontList: −adobe−courier−bold−r−*−*−14−*

For example, you can place the line above in a file called Resources, then set the XENVIRONMENT variable to point to the Resources file. Here is a c−shell example:

setenv XENVIRONMENT Resources

# *Acknowledgments*

Special thanks are in order for the following people:

*John D. Fowler Jr.* (CIC Division):
for creating the initial port to OpenGL.

*Harold Trease,*
*Manjit Sahota,*
*John McGhee,*
*Mabel Grey,*
*Paul Maudlin:*
for ideas and initial tests.

*Jeff Hinrichs,*
*Rebecca Fresquez,*
*Kevin Bolling:*
for their contributions in creating this manual.

*Glen Hansen:*
for assisting with the maintenance of GMV and this manual.

Los Alamos

NATIONAL LABORATORY

Los Alamos, New Mexico 87545